PATENT APPLICATION

OF

MICHEAL P. IVERSON AND DARON A. ISAAC

FOR

SYSTEM FOR PERFORMING COUPLED FINITE ANALYSIS

CERTIFICATE OF EXPRESS MAIL

Express Mail Mailing Label No. EL889189823US
Date of Deposit: November 7, 2001
I hereby certify that this paper or fee is being deposited with the United States Postal Service "Express Mail Post Office to Addressee" service under 37 CFR 1.10 on the date indicated above and is addressed to Box Patent Application, Assistant Commissioner for Patents, U.S. Patent and Trademark Office, Washington, D.C. 20231.

Anne K Hoole

Attorney Docket No. 1082-011

BACKGROUND OF THE INVENTION

Field of the Invention

The invention pertains to the field of software, computer systems and related methods including models that operate through the use of finite analysis, finite volume, and/or finite difference techniques.

Description of the Related Art

Finite element, finite volume, and finite difference modeling techniques can generally be described as mathematical approximations of often-complex problems that represent physical behavior. The mathematical models are useful in designing physical apparatus or systems and in predicting the behavior of existing apparatus or systems. These models use a mesh or grid that is superimposed over the system being studied to provide a plurality of cells or elements. These elements may be modeled in multiple dimensions, for example, one, two, or three dimensions. Mathematical equations that represent or approximate physical or quantitative behavior are applied to each cell with the resultant formation of a system of equations that are expressed as matrices, and that are solved using generally known techniques of linear algebra. Such mathematical techniques commonly involve iterating through a set of equations until a threshold convergence is achieved, i.e., until the difference between successive iterations through a system of mathematical approximations becomes so small that it is suitably negligible with respect to the exact or rigorous solution of the system of equations being modeled. The term "finite analysis" is hereby defined to include finite element, finite volume and

finite difference models.

A variety of patents have issued on various finite element and finite difference techniques. For example, U.S, Patent Nos. 5,956,500 and 5,901,072 pertain to a method for incorporating boundary conditions into a finite analysis model. These patents disclose generating a finite analysis model having a finite element shim interposed between a test bar and a ground, where the characteristics of the shim are selected based upon measured natural frequencies of the test bar. U.S. Patent No. 5,768,156 addresses a method of automatic mesh generation for finite analysis. The meshes are generated using whisker chords to form all-hexahedral elements. Similarly, U.S. Patent No. 5,731,817 pertains to a system that generates a hexahedron mesh and then performs finite analysis on the mesh. U.S. Patent No. 5,581,489 discloses a model generator including data input for an object to be modeled, a material information generator providing material properties for the object being modeled, a mesh processor for generating a mesh, and an output generator coupled with a finite analysis processor. U.S. Patent Nos. 5,553,206 and 5,315,537 pertain to automatic mesh generation systems.

Finite analysis programs that provide solutions to specific problems are commercially available. For example, ABAQUS is available from Hibbitt, Karlsson and Sorenson of Pawtucket, Rhode Island to model structural mechanics and nonlinear heat transfer. ANSYS is available from Ansys, Inc, of Canonsburg, PA to model structural mechanics and heat transfer. ASTMA is public domain software available from the National Aeronautics and Space Administration (NASA) that models heat transfer and

ablation. IDEAS is available from Structural Dynamics Research Corporation of Milford, Ohio to provide pre and post-processing images of the model. SINDA from SINDA, Inc. of Tempe, Arizona models heat transfer. TEX CHEM models chemical reactions and chemical equilibrium. RECESS is a program developed by Thiokol Propulsion of Brigham City, Utah to model internal ballistics. CDCA is a computational fluid dynamics program developed by Pennsylvania State University to model crack combustion where a fracture in a propellant affects burn condition. CCM is a similar computational fluid dynamics program available in the public domain, and is available from the Air Force Research Laboratory (ARFL).

Many specific examples of the need for finite analysis programs exist, for example, in the field of rocketry and missile maintenance. In fact, the commercial finite analysis programs that are mentioned above have many specific applications in this field. For example, the public domain ASTMA program and derivatives thereof can be used to model the burning away of material from a rocket engine nozzle.

A problem exists in the field of finite analysis modeling because engineering specialties do not encompass a wide array of specialized problems that are presented by complex physical situations. For example, the burning of a solid fuel rocket motor presents a multifaceted problem including structural mechanics, material properties, internal ballistics, chemical reactions, heat transfer, crack combustion, and fracture mechanics. An engineer who is modeling only one of these problems using a commercially available or proprietary finite analysis program for this purpose may require a full year just to become proficient at using the package. Such engineer is

typically not trained in more than one or two of the specialty problem areas and is often incapable of running models in areas outside his or her area of expertise. Very few, if any, engineers succeed in acquiring the training that is required to model all aspects of this problem, and a team of modelers often is required to produce modeling results through a laborious process involving the transfer of model results between different engineers and/or finite analysis codes.

It is typical in the finite analysis art that there exist separate programs to model computational fluid dynamics, structural mechanics, heat transfer, internal ballistics, etc. This segregation of problems exists, in part, due to the lack of overlap in specialty areas as described above, but it also exists because the situations encountered for actual modeling purposes are very diverse and require flexibility if the model is to have optimum results. A great deal of effort may be expended to develop a comprehensive model where the usefulness of the model diminishes with its complexity.

The foregoing problem is normally addressed by a sharing of data between engineers or engineering groups that encompass multiple specialties. This sharing of data leads to additional problems. An engineer receiving model results from another engineer for further processing does not necessarily understand the model results that he or she has received, and this circumstance can lead to computational error. For example, the preceding engineer may provide results from a less thorough model than is required for optimum results in subsequent calculations, or problems may arise through the nodes of meshes being at different locations when data is passed from a first model to a second model.

Special problems also arise when an engineer receives prior calculation results and uses them as input in a subsequent model addressing a different problem because subsequent calculation results may affect input for the prior model. For example, an internal ballistics program may be used to calculate internal pressures in a solid fuel rocket motor. These pressures are subsequently used in structural mechanics calculations where the rocket fuel deforms in a visco-elastic manner. The volume changes from the structural mechanics solution have significant effects upon the internal ballistics results which, in turn, affect the structural mechanics model. Thus, a repetitive sharing and transfer of computational results is required from successive iterations until the effects of the separate programs upon one another between different runs become negligible. Furthermore, the respective modelers may even be unaware that their individual model or an aspect of their model results can affect other models that provide results including input data for subsequent models.

Objects of the Invention

Accordingly, an object of the present invention is to provide a finite analysis modeling system that permits the user to identify a joint problem for coupled solution through the use of a graphical user interface or a scripting language.

Another object of the invention is to provide a single interface that ties or couples together a plurality of finite analysis programs for purposes of solving complex problems while maintaining flexibility.

Another object of the invention is to provide data linkages for coupling programs in a predetermined manner to facilitate coupling for the user.

Yet another object of the invention is to facilitate user-specified complex solutions without requiring the user to understand all aspects of each discipline.

These objects and advantages, as well as other objects and advantages will be apparent to those skilled in the art upon reading the attached drawings together with the accompanying specification.

BRIEF SUMMARY AND OBJECTS OF THE INVENTION

A computer system, as well as associated software and methodology are provided for obtaining a joint solution through the use of multiple finite analysis programs. The system, software and methodology tie or couple together a plurality of finite analysis programs for purposes of solving complex problems while maintaining flexibility though the use of a graphical user interface.

Preferred embodiments operate in an environment of use where there is memory storage for first and second finite analysis programs. The first finite analysis program acts upon first program input values to provide first program output values based upon the first program input values. The second finite analysis program acts upon second program input values including a first joint data set having at least a subset of the first program output values. The second program output values include a second joint data set of values that can be used as first program input values. One of the first and second finite analysis programs is a computational fluid dynamics program.

As mentioned above, the software and the method, as well as the computer system in an electronically programmed state, all utilize a graphical user interface that is

operable to identify a joint problem that both the first and second finite analysis programs can jointly and in combination solve, and to specify at least one criterion for a joint solution. Data processing linkages between the first and second programs, and the user, provide the first program input values to the first program. Embedded commands in the graphical user interface or related programs execute the first finite analysis program to obtain the first program output values including the first joint data set. Data processing linkages between the first and second programs then provide the second finite analysis program with second program input values including the first joint data set. Embedded commands in the graphical user interface, related programs, or a scripting language execute the second finite analysis program to provide second program output values including the second joint data set. Data processing linkages between the first and second programs then provide the first finite analysis program with first program input values including the second joint data set.

An aspect of the software, system, and method is that the criterion for the joint solution specified through the graphical user interface preferably includes an iterative convergence criterion for threshold convergence of the joint solution. The program instructions, as well as the methodology of operation, then include repeating the steps of providing data and executing the first and second programs until the specified threshold convergence criteria is achieved. This iteration, which is performed outside the boundaries of the first and second finite analytical programs, is performed

automatically and without user intervention after the initial data input and execution of the first and second programs.

Another aspect of the software, system and method is that the graphical user interface is optionally used to identify the joint problem by selecting the joint problem, in combination with the computational fluid dynamics problem, to preferably include an additional finite analysis problem selected from the group consisting of structural analysis problems, heat transfer problems, chemical reaction problems, chemical equilibrium problems, internal ballistics problems, and fracture mechanics problems.

A third, fourth, fifth or additional programs may be selected in like manner and placed in the coupled or joint program execution for convergence among all of the programs. For example, where a third program is selected for use in the coupled solution, the third program acts upon third program input values selected from the group consisting of first program output values, second program output values, and combinations thereof to provide third program output values. The third program output values include a third joint data set comprising input values selected from the group consisting of first program input values, second program input values and combinations thereof. The method of operation then includes executing the third program to produce the third program output values including the third joint data set. The third joint data set is provided, as needed, providing the third joint data set to the first and second finite analysis programs with corresponding input values selected from the group consisting of first program input values, second program input values and combinations thereof.

Iterative threshold convergence can then be achieved according to convergence criteria specified through the graphical user interface.

The software, system and method can be applied to a number of problems, for example, in the field of missile design and maintenance. For example, where a solid fuel rocket has maintenance operations performed on it, and these operations provide computed tomography results showing a crack in the propellant, the effect of this crack may be modeled to determine whether the crack will prevent the missile from completing its intended purpose if launched. In this case, the computational fluid dynamics program may be a crack combustion program, the system provides means for modeling crack combustion in a missile based upon computed tomography taken from a missile, and the other of the first and second programs is, by way of example, a structural analysis program.

The finite analysis may be supplemented through use of a programming language, especially an extensible object oriented scripting language that is capable of issuing calls to a variety of other programming languages. In this manner, a software screen or shell may be built to provide, for example, a function library that duplicates the GUI functionality using direct commands in the guise of scripted functions which operate across the boundaries of multiple operating systems, multiple programming languages, and/or data translations, as required to couple a plurality of programs. The scripting language preferably has looping and decisional logic capabilities giving the user infinite control over the operation of a variety of coupled finite analysis programs. The scripting

language and its associated function library may optionally be accessible from the GUI.

BRIEF DESCRIPTION OF THE DRAWINGS

Figure 1 is a schematic diagram of a preferred system embodiment according to one aspect of the invention;

Fig. 2 is a process schematic operational diagram for the system of Fig. 1;

Fig. 3 depicts a midsectional view of a rocket motor having various defects that are modeled by finite analysis causing mesh boundaries to change, in order to provide additional detail with respect aspects of the preferred method illustrated with respect to Fig. 2;

Fig. 4 is a schematic representation of menu options for a preferred graphical user interface that operates according to another aspect of the invention; and

Fig. 5 depicts a selected model that can be created using a scripting language to couple a plurality of finite analysis programs.

DETAILED DESCRIPTIPON OF THE PREFERRED EMBODIMENTS AND METHODS

According to one of the various embodiments of the invention, there is now shown a schematic computer system that is programmed to an electronically configured state through the use of software for the performance of a method, as described above. The system, software and methodology tie or couple together a plurality of finite analysis programs for purposes of solving complex problems while maintaining flexibility though the use of a graphical user interface. By "coupling" it is meant that the results

from one finite analytical model are provided to another model as input. For example, the results from a first model may be provided as input to a second model, and the results from the second model may optionally be provided as input to the first model. This process may be repeated until a user-specified convergence is achieved.

Fig. 1 depicts a schematic diagram of a computer system 100 including a central processing unit (CPU) 102 that is linked to a magnetic or optical storage medium 104, a display 106, such as a flat panel plasma display or cathode ray tube, and input/output devices 108, such as a computer keyboard, a pointing device such as a trackball or mouse, other computers, and additional magnetic or optical storage media. The hardware components 102 through 108 of system 100 may comprise those that are found in most personal computers and engineering workstations.

As shown in Fig. 1, software from storage media 104 or I/O devices 108 has been used to program a graphical user interface (GUI), which is represented by a GUI RAM block 110. The GUI 110 is used to interface with a user through I/O devices 108 to identify a plurality of programs for coupled execution. These programs are represented by RAM blocks 112-120. Block 112 includes a first finite analysis program that provides a computational fluid dynamics solution. Block 114 includes a second finite analysis program that provides a structural solution. Block 116 includes a third finite analysis program that provides a heat transfer solution. Block 118 includes a fourth finite analysis program that provides a ballistics solution. Block 120 includes a fifth finite analysis program that provides a mesh generator program. The mesh

generator program is not used in iterative convergence, but it is used to generate individual meshes for use in each of the first, second, third, fourth and fifth finite analytical programs.

The system, software and method preferably operate in an environment of use where there is memory storage for the first and second finite analysis programs corresponding to blocks 112 and 114, for example. The first finite analysis program acts upon first program input values to provide first program output values based upon the first program input values. The second finite analysis program acts upon second program input values including a first joint data set having at least a subset of the first program output values. The second program output values include a second joint data set of values that can be used as first program input values. One of the first and second finite analysis programs is a computational fluid dynamics program.

An aspect of the software, system, and method is that the criterion for the joint solution specified through the graphical user interface preferably includes an iterative convergence criterion for threshold convergence of the joint solution. The program instructions, as well as the methodology of operation, can then include repeating the steps of providing data and executing the first and second programs until the specified threshold convergence criteria is achieved. This iteration, which is performed outside the boundaries of the first and second finite analytical programs, is performed automatically and without user intervention after the initial data input and execution of the first and second programs.

Fig. 2 is a block diagram of a process 200 representing the operation of system 100. Step 202 entails using the GUI 110 to identify a problem for coupled solution between two or more of the programs shown in RAM blocks 112-118. Data linkages, as well as a sequence of operation for the respective programs to solve a coupled solution, have been previously entered by an expert or team of experts in coupling the finite analytical programs.

Data for these solutions is provided to the system 100 in step 204 where, for example, computed tomography data from missile maintenance operations may be provided as input for a structural model. Additional data including such data as materials properties; the specification of materials; boundary conditions of temperature, pressure, force, and any other useful data, is provided as needed by the specific analytical programs. A mesh generator program, such as the fifth program 120 or a plurality of such programs designed for specific applications, is also used to provide data input in the form of mesh generation.

Step 206 preferably begins once the data input of step 204 is concluded, as shown in Fig. 2, but the process 200 may also interrupt itself to ask the user for input at any time. The first program is executed in a sequence of execution designed by the expert or team of experts. The execution of the first program in step 206 provides first program output including a joint data set that may be shared, in step 208, as input data with the second finite analysis program shown in block 114. Additional data subsets may be generated and shared with any other of the finite analysis programs in blocks

116 and 118, for example. The second finite analysis program is executed in step 210 with a similar sharing of data in step 212. In step 212, however, the second program output may provide a second joint subset of data that can be used as input data for the first program once step 206 is executed again. The remaining programs are executed in a similar manner with appropriate data linkages being provided in cooperation with executable code associated with the GUI 110 so that the user does not need to specify data linkages to obtain a coupled solution.

In preferred embodiments, a portion of the data input in step 204 includes a criterion or criteria for iterative threshold convergence. For example, where the ballistics block 118 produces pressure data that modifies the program input for the structural block 114 due to the elastic deformation of rocket propellant, the initial boundary condition of pressure in the structural block 114 may be modified with time, as may the pressure conditions of the computational fluid dynamics model 112. The change in pressure from the ballistics model causes the structural volume results to change, as computed by the structural block 114. In turn, the computational results from the structural block 114 including an altered volume may be supplied as input to the computational fluid dynamics block 112 to obtain still different pressures. Both the volume results from structural block 114 and the pressure results from the computational fluid dynamics block 112 may, in turn, be supplied as input to the ballistics block 118.

In step 216, this iterative procedure continues with repetitive iterations through steps 206, 208, 210, 212, and 214 until the specified convergence criteria or a maximum number of iterations representing a probable divergent solution is achieved. For example, the user-specified convergence criteria may indicate convergence when the total pressure change for computations in the ballistics block is less than a fixed value, e.g., one-half psi between successive iterations; when the volume change in successive iterations through the structural block 114 is less than a predetermined delimiting percentage of the total volume; when calculations for the same value obtained as output from different programs match within a delimiting percentage; combinations of these examples; and any other useful convergence criteria.

Where convergence is determined to have been met in step 216, a post-processing step 218 stores the results, provides a visual display of the results on the display 106, and prints the results as needed. Step 218 also entails any other desired post-processing step.

Step 220 entails the interpolation or projection of meshes for reasons that are illustrated, by way of example, in Fig. 3. In summary, mesh boundaries frequently move as a result of physical responses to system stimuli, e.g., heat, pressure, strain, and ablation, so that the model boundaries must be adjusted due to these movements. Fig. 3 depicts a section 300 of an aging cylindrical solid fuel rocket motor. The motor includes a composite outer shell 302, a liner 304 that is used to protect the outer shell from burn-through during launch, an inner core 306 of visco-elastic rocket propellant

306, and an interior core 308 comprising a burn chamber. Conventional maintenance operations including computed tomography have diagnosed an area 310 of debonding between the propellant 306 and the liner 304. Computed tomography has also diagnosed a crack 312 that is growing in the aging rocket propellant. Finite modeling has provided a coupled solution involving internal ballistics, structural, and computational fluid dynamics to demonstrate various flow regimes including regimes 314, 316, 318, 320, and 322. A few rocket motors have been known to explode due to cracking of the propellant as shown in crack 312.

The propellant region 324 tends to deform more readily due to higher velocity downstream of crack 312, which results in a higher upstream pressure. Flow conditions around the crack 312 have a Bernouli effect that results in decreased pressure in flow regime 320 downstream of the crack 312. The regime 320 narrows the flow through regimes 316 and 318, and imparts increased velocity. Flow regime 322 is a relatively low pressure flow regime. Another potential problem is that the propellant 306 may strip away from the debond area 310 with disastrous results.

The computational fluid dynamics model begins calculation using a cylindrical mesh (not shown in Fig. 3) having an outer radius equal to the inside diameter of the propellant 306. This outer radius is shown as lines 334 and 336 in Fig. 3. As shown in Fig. 3, the computational results from a first pass iteration of the coupled programs have moved the inner diameter of the propellant 306 out to lines 326 and 328, primarily due to deformation of the propellant 306. Convergence has not yet been achieved, so it

becomes necessary to adjust the associated meshes and relevant boundary conditions to account for the deformation of propellant 306. Boundary condition projection is performed by projecting the results from the fluid mesh at lines 326 and 328 to new boundaries at lines 330 and 332. Pressure results for CFD, for example, may be projected to new boundaries of the structural mesh at lines 330 and 332.

Projecting of boundary conditions is done by locating associated positions on the complimentary mesh and extracting the necessary data to create comparable boundary conditions on the current mesh, i.e., the mesh that is receiving the boundary conditions. Usually, the associated position is perpendicular to the surface of the receiving mesh, but may be the closets point if the perpendicular is not usable. The boundary conditions are then created on the current mesh from data on the complimentary mesh.

Alternatively, where the initial pressure assumption is too high, the initial mesh for the computational fluid dynamics model may be found having a radius at lines 330 and 332. In this case, the computational results show the propellant having moved radially inboard to lines 326 and 328. In this alternative case, an identical method of projecting boundary conditions can be used.

Differences between iterative steps may require the boundary of the mesh to be modified. Large differences may require the finite element mesh to be remeshed for use in subsequent calculations. Results and boundary conditions must now be transferred to the modified mesh by interpolation for the analysis to continue. If mesh interpolation is not desired, or if the fracture geometry prevents interpolation, the user

may be prompted to assist in the generation of a new mesh over the region of concern, or an entirely new mesh may be generated over the new boundary by a mesh generation package. Techniques for automated mesh generation to discretize a model of a body are generally known. Examples of such techniques are included in the foregoing discussion of related art, e.g., U.S. Patent No. 5,729,670, which is incorporated by reference to the same extent as though fully disclosed herein.

As mentioned above, the software and the method, as well as the computer system in an electronically programmed state, all utilize a graphical user interface that is operable to identify a joint problem that both the first and second finite analysis programs can jointly and in combination solve, and to specify at least one criterion for a joint solution. Data processing linkages between the first and second programs, and the user, can provide the first program input values to the first program. Embedded commands in the graphical user interface or related programs execute the first finite analysis program to obtain the first program output values, including the first joint data set. Data processing linkages between the first and second programs then can provide the second finite analysis program with second program input values including the first joint data set. Embedded commands in the graphical user interface, related programs, or a scripting language can be used to execute the second finite analysis program to provide second program output values including the second joint data set. Data processing linkages between the first and second programs then provide the first finite analysis program with first program input values, including the second joint data set.

Fig. 4 depicts a plurality of graphical fields such as may appear, for example, on the CRT of a user who is interacting with the graphical user interface of the preferred embodiment. The graphical elements are also known as fields, and may be accessed, for example, by clicking the button of a mouse to reveal additional menu options or icons that are associated with each field.

A GUI 400 includes a **file** field 402, which includes submenu options (in bold font) including functionality to open a **new** file; **open** an existing file, **close** an active file; **read** data for input to a list-selected finite analysis program and configure that program for options and units; **write** data for input to a list-selected finite analysis program and configure that data for options and units; **translate** data from one finite analysis program to another between a list of *read from* programs and a list of *write to* programs; **save** the current file and **save as** a new filename; **print** the data representation on the screen, **print preview**, **print setup**, **copy** the data representation on the screen, and **exit** the program. In summary, these submenu options permit the user to manage and access the program data files.

A create field 404 permits the user to interact with a mesh editor to identify the location and type of mesh. The mesh is drawn interactively based upon user-specified parameters up to and including automatic mesh generation. The create field 404 includes submenu options including whether the mesh **system** is based upon rectangular, cylindrical or spherical coordinates, the reference plane that is viewed on the screen, and the starting coordinates of the mesh. A **point** option permits the user to

enter points based upon numerical coordinates or to enter points by using the mouse to select points. A **curve** may be drawn as a line, arc, circle or spline. A **surface** option permits the user to create surfaces from a list of predetermined **boundary curves**, e.g., spheres, cylinders, and planes, or by **extruding** or **rotating** curves. A **volume** option permits the user to create volumes from a list of predetermined **boundary surfaces**, e.g., spheres, cylinders, and planes, or by **extruding** or **rotating a** surface. A **node** may be created by entering the location of the desired node. The nodes may be connected by specifying **elements** that connect the nodes, as well as element geometrical behavior and material type. Nodes may be assigned positional **constraints** that may also be related to adjacent nodes and free faces. The nodes also have labels and are assigned special graphical representation, e.g., colors. Once a surface or volume has been generated, a **mesh** option permits the automatic generation of a mesh over the surface of the body or a mesh representing the volume of a body, together with entry of material properties for the mesh or elements of the mesh. Nodes, faces, elements, points, curves, surfaces, and volumes may be treated as a **group** that shares special properties. **Boundary conditions** may be defined including pressure, restraint, force, moment, temperature, convection, radiation, heat flux, point source, volume source, specie concentration, specie convection, and specie flux. New materials may be added and **material properties** may be entered to a database that tracks material properties. Similarly, **chemical species** may be added and **chemical reactions** may be entered into an interpretive equation editor. Additional tools for displaying the

meshes include a plurality of CAD related menu options, e.g., to view a mesh or access geometry creation functions. In summary, the **create** field provides an interactive mechanism for creating meshes, creating boundary conditions, and assigning chemical and material properties to components of the meshes.

An edit field 406 permits the user to modify the **work plane** that is represented on the CRT for viewing. The orientation of the work plane may be designated by coordinate entry, and the character of the work plane can be designated as rectangular or cylindrical. The color and appearance of mesh or body components may be changed, e.g., as by changing the color of **points**, **curves**, **surfaces**, **volumes**, **nodes**, and **elements**. Chemical reaction data may be edited. In summary, the **edit** field permits the user to change the appearance of an existing mesh without deleting elements of the mesh or body.

A **delete** field 408 has the same list of submenu options as does the **create** field, except the submenu options are used to delete elements, as opposed to creating them. Additionally, boundary conditions may be deleted in sets of related boundary conditions. A **list** field 410 is used to provide a list of items encompassing the same menu options that exist for the **delete** field.

A **post-processing field** 412 contains submenu options including **superposition** of calculation results on the mesh; interpolation of results from one mesh to another as specified by model names for the meshes; interpolation to **nodes**, or **centroids**, and to average results; **deformed geometry** may be shown to compare

the degree of deformation in a system component as brought about by the predictive model results; contour plot, vector plots, and X-Y plots of predetermined data may also be obtained.

A **model** field 414 permits the user to identify a selected model for the coupled solution of a joint problem. A **tools** field includes a **coincident node check** that identified nodes having locations within a predetermined or user-specified tolerance of one another. An **element distortion check** option permits the user to specify which control measures for distortion the system will consider and to input the desired control values including control measures for Gauss point distortion, aspect ratio, area ratio, quad angle (90°+/-), triangle angle (45° +/-), and **element** warping.

A **view** field 418 has submenu options including selected adjustments to **display settings** including color representation of diagram components on the display; **blanking** and **unblanking** of points, curves surfaces, nodes, volumes and elements on the display; **dynamic view** including pan, magnify, rotate, 3D rotate, and zoom options; **reset** of the dynamic model changes including a return to the originally depicted conditions; **autoscale** in which the scale of the model representation on the display is adjusted to reveal the entire model in optimum context; **view orientation** in which the display on the screen is changed to represent an eye orientation form a user-entered set of coordinates, rotation of the screen perspective a specified distance about a user-selected coordinate, and continuing rotation of the model representation about a user-selected axis.

The help field provides hyperlink access to user information, such as the information that is shown in the attached appendix.

Fig. 5 depicts a preferred process 500 for solving a joint problem of a type that may be selected from a menu option using the GUI 400, namely, a structural analysis ballistics problem. Geometry data involving the interior of a missile is obtained using conventional computed tomography techniques for nondestructive evaluation (NDE) of the motor core. This data is provided in step 502 as a body for modeling situations of the type shown in Fig. 3, for example. Step 504 entails automated meshing and may be performed using any number of conventional automatic meshing programs. In step 506, a processor receives user-selected input data and executes program instructions for the first analysis program, which in this case happens to be a structural analysis program modeling time dependent nonlinear visco-elastic (NLVE) deformation of the rocket motor as the interior propellant burns. Structural integrity is checked in step 508 to assure that the motor has not exceeded design specifications or a failure point.

The NLVE output includes deformations, stresses and strains, which are provided as second program input values to a second program, which comprises a fracture analysis or fracture propagation program that is used in step 510. The output from the fracture analysis program includes deformation and fracture propagation with automated meshing of the propagation-deformed system. A burnback analysis is executed in step 512 including an approximation of the burning surface area that is modified or deformed through burning in the rocket motor core. The burnback analysis

output includes a time-dependent surface area, and a new mesh may be generated over this area.

The new surface area and mesh is supplied to a computational fluid dynamics and/or ballistics program as program input data in step 514. Iteration continues along loop 516 until convergence is achieved and performance data is supplied in step 518. The computational fluid dynamics program that is used in step 514 may be configured to provide a three dimensional transient solution that has not been previously done in the art. For example, a special scripting language may be used to provide threshold convergence of the solution at small time steps with sequential performance data being provided in step 516 for each time step. Computations of this type may require several CPU weeks to complete, even where the processor is operating in the gigahertz range of clock speeds, and the manual intervention that would have been required to couple the programs for this type of solution was simply impossible using prior methods.

Other coupled CFD solutions include steady state flow for 1D, 2D and 3D systems in combination with elastic structural deformation or nonlinear visco-elastic structural deformation; quasi-steady state flow in combination with crack propagation; transient 1D, 2D and 3D flow in combination with elastic structural deformation or nonlinear visco-elastic structural deformation; and transient flow with crack propagation.

It should be noted that any one of the programs which is executed in loop 516 can be a first or second program of the type described herein. The order of execution of the first and second program can be any order because the first and second designation

serves merely to distinguish one program from another and does not relate to any specific order of program execution unless otherwise specified. Thus, the computational fluid dynamics program may be executed fourth in order, and this order of execution is still consistent with describing the program as a first or second program. Additionally, even though the respective programs of steps 506-514 are different programs and are not simultaneously executed by a processor, although they may be simultaneously executed in a distributed processing or multitasking environment, these program are said to *jointly and in combination* provide a solution to the joint problem of structural ballistics analysis because they cooperate in an iterative scheme for a joint solution and/or the programs share data with one another even if no repeat iteration is required.

Prior mention has been made of a scripting language. The scripting language permits advanced users or system experts to issue program commands that are comparable in analogy to function calls from an object-oriented programming language. A particularly preferred scripting language is Python, which is a copyrighted but freely usable and distributable product, even for commercial use, and is available from PythonLabs at www.python.org.

The Python scripting language is often compared to other object-oriented programming languages including Tcl, Perl, Scheme or Java, which may provide object oriented substitutes for Python. According to a presently contemplated but merely illustrative embodiment of the invention, the executable code supporting the GUI 400 is

written in Python scripting language by an expert in one or more of the finite element analytical programs. An *expert* is hereby defined as a person who has at least five hundred hours of training and program use in a particular finite analysis program, and this time of use is preferably more than one thousand hours.

An additional GUI element preferably makes available the scripting language to ordinary users who may alter copies of expert-preprogrammed code or write their own code. On networked systems, these user-defined scripting packages may be shared among all users, subject to an expert review and approval process. A Python module was created as an alternative scripted interface comparable to GUI 400, to support geometry creation, finite element models for solving joint problems, data file interfaces and data linkages between different finite analytical programs, and post processing activities. An example of Python code for a data file translation script between two finite analysis programs is:

```
import FEM
model=doc.NewModel('PlaneE1')
model.RdIdeasMS('RWIdeas1.unv')
model.WrAbaqus('../Work/PlaneE1inp')
```

A preprogrammed library of basic functions that duplicate the GUOI functionality is provided for user-specified execution in the Python scripting language, which may issue calls to C++ language functions and other languages. The function library may be accessed directly to provide the user with the ability to perform automated calculations

without having to interact with the GUI, however, the library is preferably accessed through use of a submenuing or execute option permitting a user to access and/or reprogram the library functions. Classes of functions include functions that create new documents, open existing documents, save documents, create curves, surfaces, volumes, create boundary conditions, interpolate, read and write files, translate data between applications, as well as any other activity that has been previously described as being practicable through the use of GUI 400. In part because the Python code is extensible to other languages, Python provides tremendous power and versatility including looping and decisional logic capabilities.

Applicant hereby incorporates-by-reference Appendix I (printouts from an interactive online user's manual for the FEM Builder), Appendix II (a compact disk in duplicate comprising a computer program listing appendix containing computer software according to a presently preferred system and method according to the invention), and Appendix III (a computer program listing appendix containing the file and directory names for the files and directories on the compact disk comprising Appendix II).

The foregoing discussion provides examples that are intended to operate by way of example, not by limitation. There may be additions or changes to the processes and programs described above that fall within the scope and spirit of the invention. For example, any processing system may be used to execute the program instructions including systems having distributed processing networks, distributed data storage, and multiple I/O devices, in addition to the system shown in Fig. 2. Accordingly, the inventors hereby state their intention to rely upon the Doctrine of Equivalents to protect their full rights in the invention.

APPENDIX I

The following pages are printouts from an interactive online user's manual for the FEM Builder, which is a computer program that has been built by engineers at Thiokol Propulsion of Brigham City, Utah. The FEM Builder program operates according to the principles of the invention to accomplish coupling of multiple finite analysis programs through the use of a graphical user interface and/or a scripting language.

FEM Builder User's Manual

FEM Builder is a PC program written at Thiokol Corporation to provide additional Finite Element Modeling (FEM) tools for building and analyzing finite element models, and to provide interfaces between FEM systems and analysis programs.

Table of Contents

Introduction

File
- Abaqus
  - Read Results File
  - Read Input File
  - Write Input File
- Ansys
  - Read Results File
  - Read Coded Database File
  - Write Coded Database File
- Aschar
  - Read Aschar Output File
- GridGen
  - Read GridGen File
- Ideas Master Series
  - Read Universal File
  - Write Universal File
- Patran
  - Read Neutral File
  - Write Neutral File
  - Write Results File
- Sinda
  - Write Sinda input File
- TexChem
  - Read TexChem Files
  - Write TexChem Files
- Material Properties
  - Read Material Property File
  - Write Material Property File

Create
- Geometry (coordinate systems, points, curves, surfaces, and volumes)
- FEM Entities (nodes, elements, constraint equations, groups, and boundary conditions)
- Mesh Generation
- Material Properties
- Chemical Species and Reactions

Edit
- Geometry (work plane, points, curves, surfaces, and volumes)
- FEM Entities (nodes, elements)
- Material Properties

Delete

List

Post-processing
- Result Calculation (Superposition, Interpolation)
- Result Display (Deformed Geometry, Contour Plots, Vector Plots, XY Plots)

Model Information

Tools

View

Window

Appendix

- Element Library
- Abaqus Element Support
- Ansys Element Support
- Material Property Support
- Selection (Systems, Point, Curve, Surface, Volume, Node, Element, Face, Groups, Material Properties, BCs, Results)
- Python Module
- Work Directory

Introduction

FEM Builder is an interactive program written by Thiokol Propulsion to provide Finite Element Modeling (FEM) tools for building and analyzing finite element models, and to provide interfaces between FEM systems and analysis programs.

Fem Builder has a user interface similar to most PC programs with a menu bar, a toolbar, and a status bar. The program allows users to have one or more documents open at the same time. A document can contain one or more finite element models. Each document can have multiple views.

File

File functions allow the user to **Open**, **Close**, and **Save** FEM Builder files. **Read** and **Write** functions are provided to read files from and write files to various analysis programs. Other interfaces are provided to read and write material property files. The **Translation** option translates data from one file type to another without saving that information in a FEM Builder document, which can save time if no other FEM Builder functions are to be performed. File functions also allow the user to **Print** the current view, or **Copy** the view to the clipboard for pasting in applications like Microsoft Word®. The recent file list contains the most recently used FEM Builder files.

Create

Creation functions allow the user to create geometry (coordinate systems, points, curves, surfaces, and volumes), FEM entities (nodes, elements, constraint equations, groups, and boundary conditions), perform mesh generation, and define material properties, chemical species and chemical reactions.

Edit

Edit functions allow the user to edit FEM entities. Functions exist to edit geometry, material properties, and element orientations.

Delete

Delete functions delete selected FEM entities from the current model.

List

List functions write selected information to the List Box.

Post-Processing

Post-processing functions include result superposition, and result interpolation, deformed geometry, contour plots, vector plots, and XY plots.

Model Information

Most FEM Builder functions operate on one finite element model, the current model. The Model Information dialog allows the user to set the current model, the model name, and the model title. The dialog also lists other model-related information.

Tools

Tools include coincident node check, element distortion check, and customizable default options. The default options are entity picking, work plane use, element defaults, subdivision, and units.

View

The display settings are used to set the visibility and defaults of geometry entities, fem entities, and the view header. The view orientation can be specified directly via keyboard input. The view origin, size and

orientation can also be modified dynamically using dynamic view. Status flags indicate the visibility of the toolbar, status bar, and list box.

Window

These window functions are the same functions found in many other PC programs. The **New Window** function creates a new view of the model that can be scaled, rotated, and relocated independently of any other view. The **Cascade**, **Tile**, and **Arrange Icon** options all rearrange existing views.

Help

The help option provides access to program version information.

Program Defaults

The coordinate system used by FEM Builder is the same used by most analysis programs. Interpretation of coordinates depends on the element being referenced. For example, for plane elements coordinates 1,2 are X,Y, for axisymmetric elements coordinates 1,2 are R,Z, and for solid elements coordinates 1,2,3 are X,Y,Z.

Default model units are Inches, pound mass, pound force, second, and degrees Fahrenheit.

Return to Table of Contents

ABAQUS

ABAQUS is a finite element code primarily used for structural analysis. FEM Builder supports model definition, boundary condition creation, and specification of material properties. Interfaces for reading results files, reading input files, and writing input files are described in the following sections. This interface was written to support ABAQUS 5.8-18.

ABAQUS Read Results File Interface

This interface reads ABAQUS file output files. Nodes, elements, groups, and analysis results are extracted from the file data. Appendix - ABAQUS Support: Table 1 lists supported ABAQUS element types. Groups with group names of the form MIDn will be used to set element material Ids. The following analysis results are supported.

Nodal Results

Description	Record Key
Displacements	101
Reaction forces	104
Nodal temperatures	201

Centroidal and Element Nodal Results

Description	Record Key
Stress	11
Strain	21
Elastic strain	25
Inelastic strain	24
Plastic strain	22
Creep strain	23
Thermal strain	88
Nominal strain	90
Logarithmic strain	89
Principal stresses	401
Principal strains	403
Principal Elastic strain	408
Principal Inelastic strain	409
Principal Plastic strain	411
Principal Creep strain	412
Principal Thermal strain	410
Principal Nominal strain	404
Principal Logarithmic strain	405
Temperature	2
Strain energy density	14
Stress Invarients	12
Shell section forces	13
Shell section strains	29
State variables	5

Top

ABAQUS Read Input File Interface

This interface reads ABAQUS input files and extracts FEM data from supported keyword card sections.

Keyword card support

Keyword	Supporting keywords
*Heading	
*Include	
*System	
*Node	
*Element	
*Nset	
*ElSet	
*Transform	
*Equation	
*Beam Section	
*Shell Section	
*Solid Section	
*Material	
	*Conductivity
	*Density
	*Elastic
	*Expansion
	*Specific Heat
*Orientation	
*Boundary	
*Step	
	*Boundary
	*CLoad
	*DLoad
*End	

When these keyword cards are encountered, notification is sent to the list file. These keywords are not case sensitive.

Keyword card support description

***HEADING**

The heading text will replace the existing model title, if any.

***Include**

*Include cards may be used to reference files containing additional input. Include files may have references to other include files. If the attempt to open the specified file fails, input processing **will** continue. Notification of the open failure will be sent to the list file. The most common cause of open failures is due to errors in the path specification that is part of the file name specification.

***System**

*System defined coordinate systems are used to transform the coordinates of nodes in following *Node sections into global coordinates. Defined systems will be stored in the FEM Builder database. System labels will be generated.

***Node**

The **NSet**, and **System** parameters are supported. The **File** parameter is not supported. If a **System** parameter is defined the coordinates are transformed to a Cartesian system immediately. If a ***System** section preceded this section, the coordinates are then transformed via that system into global coordinates.

***Element**
The **Type** and **ElSet** parameters are supported. The **File** and **Input** parameters are not supported. If the specified element type is not recognized, notification will be sent to the list file and the rest of the section will be skipped. Appendix - ABAQUS Support: Table 1 lists supported ABAQUS element types. If the **ElSet** name is of the form MIDn, the elements will be assigned n as the material ID.

***Nset**
The **NSet** and **Generate** parameters are supported. The **Unsorted** parameter, if defined, will be ignored. The **ElSet** parameter is not supported, and if defined, notification will be sent to the list file and the rest of the section will be skipped.

***ElSet**
The **ElSet** and **Generate** parameters are supported.

***Transform**
The **NSet** and **Type** parameters are supported. A coordinate system will be created as defined by the input. Nodes in the referenced NSet will be modified to reference the created displacement coordinate system. If the NSet name is of the form Tran#, the # is assumed to be an undefined coordinate system number, otherwise the system label will be generated.

***Equation**
The **Input** parameter is supported.

***Beam Section**
This section is only used to tie the elements referenced by the **ElSet** parameter to the material referenced by the **Material** parameter. The other data defined in this section are ignored.

***Solid Section and *Shell Section**
These sections are used to tie the elements referenced by the **ElSet** parameter to the material referenced by the **Material** parameter or to the composite layup defined when the **Composite** parameter is defined. The **Orientation** parameter, if defined, is used to orient the elements.

***Material**
The **Name** parameter is supported.

***Conductivity**
The **Type** parameter is supported. The **Dependencies** parameter is ignored.

***Density**
The **Dependencies** parameter is ignored.

***Elastic**
The **Type** parameter is supported. The **Dependencies** parameter is ignored.

***Expansion**
The **Type** and **Zero** parameters are supported. The **Dependencies** and **Pore Fluid** parameters are ignored.

***Specific Heat**
The **Dependencies** parameter is ignored.

***Orientation**
The **Name**, **System**, and **Definition = Coordinates** parameters are supported. Other values of the Definition parameter are not supported, and if encountered, notification will be sent to the list file and the rest of the section will be skipped.

***Boundary**
Restraints defined preceding the ***Step** card will be assigned a set ID of 1. Node sets may be referenced, and **XSYMM, YSYMM, ZSYMM, XASYMM, YASYMM, ZASYMM, ENCASTRE**, and **PINNED** "Type" specifications may be used.

***Step**
When this card is encountered, notification is sent to the list file. The load set ID has an initial value of 1. When the *End Step card is encountered, the load set ID will be incremented.

***Boundary**
Restraints defined will be assigned the current load set ID. Node sets may be referenced, and **XSYMM, YSYMM, ZSYMM, XASYMM, YASYMM, ZASYMM, ENCASTRE**, and **PINNED** "Type" specifications may be used.

***CLoad**
Concentrated loads defined will be assigned the current load set ID. All parameters are ignored. Node sets may be referenced. Temperature restraints and forces are supported.

***DLoad**
Distributed loads defined will be assigned the current load set ID. All parameters are ignored. Node sets may be referenced. Pressures are supported.

***End**
When the *End card is encountered, the load set ID will be incremented.

Label generation

When an entity is created with no defined label, such as for the *System card sections, the "unknown" label will initially be assigned. After reading the entire input file, these labels will be reassigned to a generated value. Generated values will increment from an initial value computed as follows:

Initial value = 1+10^(Log10(Previous maximum)+1)

Section cards

Abaqus uses section cards to associate elements with material properties. FEM Builder will use the section information to associate element material Ids with materials. When an ElSet name of the form MIDn is not on the *Element card elements will not have an assigned material ID. If elements referenced by section cards do not have an assigned material ID, a material ID will be generated.

Top

ABAQUS Write Input File Interface

This interface writes ABAQUS input. Nodes, elements, constraint equations, node transforms, groups, element section, orientation, material properties, step information, boundary conditions, and analysis results for initial conditions are written to the file. The interface also allows data of various types to be written to separate files.

Nodes

The interface writes node label, and XY, RZ, or XYZ coordinates.

Elements

The Fem Builder supported element types (geometry and analysis types) are found in Appendix - Element Library: Table 1 and Appendix - Element Library: Table 2 respectively. Appendix - ABAQUS Support: Table 1 lists each Fem Builder geometry and analysis type and the supported ABAQUS element type that it

will be assigned as the input file is written. As elements are output ELSets are defined for each material. Do not delete or rename these ELSets or FEM Builder will not be able to determine the proper element MID when reading result files.

Equations

Constraint equations are written as *Equations.

Transforms

NSets and Transform commands are created for nodes with local displacement coordinate systems.

Groups

NSets and ELSets are created for each node and element group.

Section Data

Section cards and corresponding ELSets are generated for each unique combination of element type (beam, shell, or solid), material id, property id, and element/material orientation. Multiple sections may reference the same material.

Material Properties

The following table lists the ABAQUS supported material properties and their supported counterparts in Fem Builder.

ABAQUS	Fem Builder
Elastic, Type=Iso Elastic, Type= Engineering Constants Elastic, Type=Aniso	Elastic Modulus – Isotropic Elastic Modulus - Orthotropic Elastic Modulus – Anisotropic
Expansion, Type=Iso Expansion, Type=Ortho Expansion, Type=Aniso	Expansion – Isotropic Expansion – Orthotropic Expansion – Anisotropic
Conductivity, Type=Iso Conductivity, Type=Ortho Conductivity, Type=Aniso	Conductivity – Isotropic Conductivity – Orthotropic Conductivity – Anisotropic
Density	Density
Specific Heat	Specific Heat – CP

Initial Conditions

Nodal temperatures and element centroidal stresses may be output as initial conditions. If the selected temperatures are centroidal temperatures, nodal averaging will be performed. If the selected stresses are element nodal stresses, stresses will be interpolated to element centroids using the element shape functions.

Loads and Boundary Conditions

Fem Builder outputs restraints (Boundary), forces/moments (CLoad), and pressure (DLoad) boundary conditions.

Top

Return to Table of Contents

Appendix - ABAQUS Element Type Support

Table 1 lists each supported ABAQUS element type and the geometry type and analysis type that it is assigned when reading into Fem Builder. It also lists the ABAQUS element types that are written to ABAQUS from Fem Builder. ABAQUS element types not directly supported on output will be mapped to the element type listed in the ABAQUS output column.

Table 1: ABAQUS element type support

ABAQUS Element		FEM Builder Element		Output
Name	***Description***	***Geom***	***Anal***	***Name***
B22	3 Node quadratic 2D beam	2	25	B22
B22H	3 Node quadratic 2D beam, hybrid	2	25	B22H
B23	2 Node cubic 2D beam	1	25	B23
B23H	2 Node cubic 2D beam, hybrid	1	25	B23H
B32	3 Node quadratic 3D beam	2	26	B32
B32H	3 Node quadratic 3D beam, hybrid	2	26	B32H
B33	2 Node cubic 3D beam	1	26	B33
B33H	2 Node cubic 3D beam, hybrid	1	26	B33H
C3D4	4 Node linear tetrahedron	9	60	C3D4
C3D4H	4 Node linear tetrahedron, hybrid	9	60	C3D4H
C3D6	6 Node linear wedge	12	60	C3D6
C3D6H	6 Node linear wedge, hybrid	12	60	C3D6H
C3D8	8 Node linear brick	15	60	C3D8
C3D8H	8 Node linear brick, hybrid	15	60	C3D8H
C3D8R	8 Node linear brick, reduced	15	60	C3D8R
C3D8RH	8 Node linear brick, reduced, hybrid	15	60	C3D8RH
C3D10	10 Node quadratic tetrahedron	10	60	C3D10
C3D10H	10 Node quadratic tetrahedron, hybrid	10	60	C3D10H
C3D15	15 Node quadratic wedge	13	60	C3D15
C3D15H	15 Node quadratic wedge, hybrid	13	60	C3D15H
C3D20	20 Node quadratic brick	16	60	C3D20
C3D20H	20 Node quadratic brick, hybrid	16	60	C3D20H
C3D20R	20 Node quadratic brick, reduced	16	60	C3D20R
C3D20RH	20 Node quadratic brick, reduced, hybrid	16	60	C3D20RH
CAX3	3 Node linear axisymmetric triangle	3	43	CAX3
CAX3H	3 Node linear axisymmetric triangle, hybrid	3	43	CAX3H
CAX4	4 Node linear axisymmetric quad	6	43	CAX4
CAX4H	4 Node linear axisymmetric quad, hybrid	6	43	CAX4H
CAX4I	4 Node linear axisymmetric quad, incompatible	6	43	CAX4H
CAX4IH	4 Node linear axisymmetric quad, incompatible, hybrid	6	43	CAX4H
CAX4R	4 Node linear axisymmetric quad, reduced	6	43	CAX4R
CAX4RH	4 Node linear axisymmetric quad, reduced, hybrid	6	43	CAX4RH
CAX6	6 Node quadratic axisymmetric triangle	4	43	CAX6
CAX6H	6 Node quadratic axisymmetric triangle, hybrid	4	43	CAX6H
CAX8	8 Node quadratic axisymmetric quad	7	43	CAX8
CAX8H	8 Node quadratic axisymmetric quad, hybrid	7	43	CAX8H
CAX8R	8 Node quadratic axisymmetric quad, reduced	7	43	CAX8R
CAX8RH	8 Node quadratic axisymmetric quad, reduced, hybrid	7	43	CAX8RH

CAXA4n	4 Node linear asymmetric quad, n modes	6	44	CAXA4n
CAXA4Hn	4 Node linear asymmetric quad, hybrid, n modes	6	44	CAXA4Hn
CAXA4Rn	4 Node linear asymmetric quad, reduced, n modes	6	44	CAXA4Rn
CAXA4RHn	4 Node linear asymmetric quad, reduced, hybrid, n modes	6	44	CAXA4RHn
CAXA8n	8 Node quadratic asymmetric quad, n modes	7	44	CAXA8n
CAXA8Hn	8 Node quadratic asymmetric quad, hybrid, n modes	7	44	CAXA8Hn
CAXA8Rn	8 Node quadratic asymmetric quad, reduced, n modes	7	44	CAXA8Rn
CAXA8RHn	8 Node quadratic asymmetric quad, reduced, hybrid, n modes	7	44	CAXA8RHn
CGAX3	3 Node generalized linear axisymmetric triangle	3	43	CGAX3
CGAX3H	3 Node generalized linear axisymmetric triangle, hybrid	3	43	CGAX3H
CGAX4	4 Node generalized linear axisymmetric quad	6	43	CGAX4
CGAX4H	4 Node generalized linear axisymmetric quad, hybrid	6	43	CGAX4H
CGAX4R	4 Node generalized linear axisymmetric quad, reduced	6	43	CGAX4R
CGAX4RH	4 Node generalized linear axisymmetric quad, reduced, hybrid	6	43	CGAX4RH
CGAX6	6 Node generalized quadratic axisymmetric triangle	4	43	CGAX6
CGAX6H	6 Node generalized quadratic axisymmetric triangle, hybrid	4	43	CGAX6H
CGAX8	8 Node generalized quadratic axisymmetric quad	7	43	CGAX8
CGAX8H	8 Node generalized quadratic axisymmetric quad, hybrid	7	43	CGAX8H
CGAX8R	8 Node generalized quadratic axisymmetric quad, reduced	7	43	CGAX8R
CGAX8RH	8 Node generalized quadratic axisymmetric quad, reduced, hybrid	7	43	CGAX8RH
CPE3	3 Node linear plane strain triangle	3	41	CPE3
CPE3H	3 Node linear plane strain triangle, hybrid	3	41	CPE3H
CPE4	4 Node linear plane strain quad	6	41	CPE4
CPE4H	4 Node linear plane strain quad, hybrid	6	41	CPE4H
CPE4R	4 Node linear plane strain quad, reduced	6	41	CPE4R
CPE4RH	4 Node linear plane strain quad, reduced, hybrid	6	41	CPE4RH
CPE6	6 Node quadratic plane strain triangle	4	41	CPE6
CPE6H	6 Node quadratic plane strain triangle, hybrid	4	41	CPE6H
CPE8	8 Node quadratic plane strain quad	7	41	CPE8
CPE8H	8 Node quadratic plane strain quad, hybrid	7	41	CPE8H
CPE8R	8 Node quadratic plane strain quad, reduced	7	41	CPE8R
CPE8RH	8 Node quadratic plane strain quad, reduced, hybrid	7	41	CPE8RH
CPS3	3 Node linear plane stress triangle	3	40	CPS3
CPS4	4 Node linear plane stress quad	6	40	CPS4
CPS6	6 Node quadratic plane stress triangle	4	40	CPS6
CPS8	6 Node quadratic plane stress quad	7	40	CPS8
DASHPOT1	1 Node to ground dashpot	0	2	DASHPOT1
DASHPOTA	2 Node dashpot	1	22	DASHPOTA
DC1D2	2 Node heat transfer link	1	29	DC1D2
DC1D3	3 Node heat transfer link	2	29	DC1D3
DC2D3	3 Node linear heat transfer triangle	3	48	DC2D3
DC2D4	4 Node linear heat transfer quad	6	48	DC2D4
DC2D6	6 Node quadratic heat transfer triangle	4	48	DC2D6
DC2D8	8 Node quadratic heat transfer quad	7	48	DC2D8
DC3D4	4 Node linear heat transfer tetrahedron	9	61	DC3D4
DC3D6	6 Node linear heat transfer wedge	12	61	DC3D6
DC3D8	8 Node linear heat transfer brick	15	61	DC3D8
DC3D10	10 Node quadratic heat transfer tetrahedron	10	61	DC3D10
DC3D15	15 Node quadratic heat transfer wedge	13	61	DC3D15
DC3D20	20 Node quadratic heat transfer brick	16	61	DC3D20
DCAX3	3 Node linear axisymmetric heat transfer triangle	3	49	DCAX3
DCAX4	4 Node linear axisymmetric heat transfer quad	6	49	DCAX4
DCAX6	6 Node quadratic axisymmetric heat transfer triangle	4	49	DCAX6

DCAX8	8 Node quadratic axisymmetric heat transfer quad	7	49	DCAX8
DINTER1	1 Node per side heat transfer interface	30	85	DINTER1
DINTER2	2 Node per side heat transfer interface	31	85	DINTER2
DINTER2A	2 Node per side axisymmetric heat transfer interface	31	86	DINTER2A
DINTER3	3 Node per side heat transfer interface	32	85	DINTER3
DINTER3A	3 Node per side axisymmetric heat transfer interface	32	86	DINTER3A
DINTER4	4 Node per face heat transfer interface	34	85	DINTER4
DINTER8	8 Node per face heat transfer interface	35	85	DINTER8
DSAX1	2 Node axisymmetric heat transfer shell	1	30	DSAX1
DSAX2	3 Node axisymmetric heat transfer shell	2	30	DSAX2
GAPUNI	2 Node gap	30	83	GAPUNI
INTER2	2 Node per side interface	31	83	INTER2
INTER2A	2 Node per side axisymmetric interface	31	84	INTER2A
INTER3	3 Node per side interface	32	83	INTER3
INTER3A	3 Node per side axisymmetric interface	32	84	INTER3A
INTER4	4 Node per face interface	34	83	INTER4
INTER8	8 Node per face interface	35	83	INTER8
INTER9	9 Node per face interface	36	83	INTER9
IRS21A	3 Node axisymmetric rigid surface	41	28	IRS21A
ISL21A	3 Node axisymmetric slide line	1	28	ISL21A
M3D3	3 Node linear membrane triangle	3	45	M3D3
M3D4	4 Node linear membrane quad	6	45	M3D4
M3D4R	4 Node linear membrane quad, reduced	6	45	M3D4R
M3D6	6 Node quadratic membrane triangle	4	45	M3D6
M3D8	8 Node quadratic membrane quad	7	45	M3D8
M3D8R	8 Node quadratic membrane quad, reduced	7	45	M3D8R
MASS	1 Node point mass	0	3	MASS
S4R5	4 Node linear curved thin shell, reduced	6	46	S4R5
S8R5	8 Node quadratic curved thin shell, reduced	7	46	S8R5
SAX1	2 Node linear axisymmetric shell	1	27	SAX1
SAX2	3 Node quadratic axisymmetric shell	2	27	SAX2
SAXA1n	2 Node linear asymmetric shell, n modes	1	28	SAXA1n
SAXA2n	3 Node quadratic asymmetric shell, n modes	2	28	SAXA2n
SPRING1	1 Node to ground spring	0	0	SPRING1
SPRINGA	2 Node to node spring	1	20	SPRINGA
STRI35	3 Node linear curved thin shell	3	46	STRI35
STRI65	6 Node quadratic curved thin shell	4	46	STRI65
T2D2	2 Node linear 2D truss	1	23	T2D2
T2D2H	2 Node linear 2D truss, hybrid	1	23	T2D2H
T2D3	3 Node quadratic 2D truss	2	23	T2D3
T2D3H	3 Node quadratic 2D truss, hybrid	2	23	T2D3H
T3D2	2 Node linear 3D truss	1	24	T3D2
T3D2H	2 Node linear 3D truss, hybrid	1	24	T3D2H
T3D3	3 Node quadratic 3D truss	2	24	T3D3
T3D3H	3 Node quadratic 3D truss, hybrid	2	24	T3D3H

Return to Table of Contents

Appendix - ABAQUS Element Type Support

Table 1 lists each supported ABAQUS element type and the geometry type and analysis type that it is assigned when reading into Fem Builder. It also lists the ABAQUS element types that are written to ABAQUS from Fem Builder. ABAQUS element types not directly supported on output will be mapped to the element type listed in the ABAQUS output column.

Table 1: ABAQUS element type support

ABAQUS Element		FEM Builder Element		Output
Name	***Description***	***Geom***	***Anal***	***Name***
B22	3 Node quadratic 2D beam	2	25	B22
B22H	3 Node quadratic 2D beam, hybrid	2	25	B22H
B23	2 Node cubic 2D beam	1	25	B23
B23H	2 Node cubic 2D beam, hybrid	1	25	B23H
B32	3 Node quadratic 3D beam	2	26	B32
B32H	3 Node quadratic 3D beam, hybrid	2	26	B32H
B33	2 Node cubic 3D beam	1	26	B33
B33H	2 Node cubic 3D beam, hybrid	1	26	B33H
C3D4	4 Node linear tetrahedron	9	60	C3D4
C3D4H	4 Node linear tetrahedron, hybrid	9	60	C3D4H
C3D6	6 Node linear wedge	12	60	C3D6
C3D6H	6 Node linear wedge, hybrid	12	60	C3D6H
C3D8	8 Node linear brick	15	60	C3D8
C3D8H	8 Node linear brick, hybrid	15	60	C3D8H
C3D8R	8 Node linear brick, reduced	15	60	C3D8R
C3D8RH	8 Node linear brick, reduced, hybrid	15	60	C3D8RH
C3D10	10 Node quadratic tetrahedron	10	60	C3D10
C3D10H	10 Node quadratic tetrahedron, hybrid	10	60	C3D10H
C3D15	15 Node quadratic wedge	13	60	C3D15
C3D15H	15 Node quadratic wedge, hybrid	13	60	C3D15H
C3D20	20 Node quadratic brick	16	60	C3D20
C3D20H	20 Node quadratic brick, hybrid	16	60	C3D20H
C3D20R	20 Node quadratic brick, reduced	16	60	C3D20R
C3D20RH	20 Node quadratic brick, reduced, hybrid	16	60	C3D20RH
CAX3	3 Node linear axisymmetric triangle	3	43	CAX3
CAX3H	3 Node linear axisymmetric triangle, hybrid	3	43	CAX3H
CAX4	4 Node linear axisymmetric quad	6	43	CAX4
CAX4H	4 Node linear axisymmetric quad, hybrid	6	43	CAX4H
CAX4I	4 Node linear axisymmetric quad, incompatible	6	43	CAX4H
CAX4IH	4 Node linear axisymmetric quad, incompatible, hybrid	6	43	CAX4H
CAX4R	4 Node linear axisymmetric quad, reduced	6	43	CAX4R
CAX4RH	4 Node linear axisymmetric quad, reduced, hybrid	6	43	CAX4RH
CAX6	6 Node quadratic axisymmetric triangle	4	43	CAX6
CAX6H	6 Node quadratic axisymmetric triangle, hybrid	4	43	CAX6H
CAX8	8 Node quadratic axisymmetric quad	7	43	CAX8
CAX8H	8 Node quadratic axisymmetric quad, hybrid	7	43	CAX8H
CAX8R	8 Node quadratic axisymmetric quad, reduced	7	43	CAX8R
CAX8RH	8 Node quadratic axisymmetric quad, reduced, hybrid	7	43	CAX8RH

CAXA4n	4 Node linear asymmetric quad, n modes	6	44	CAXA4n
CAXA4Hn	4 Node linear asymmetric quad, hybrid, n modes	6	44	CAXA4Hn
CAXA4Rn	4 Node linear asymmetric quad, reduced, n modes	6	44	CAXA4Rn
CAXA4RHn	4 Node linear asymmetric quad, reduced, hybrid, n modes	6	44	CAXA4RHn
CAXA8n	8 Node quadratic asymmetric quad, n modes	7	44	CAXA8n
CAXA8Hn	8 Node quadratic asymmetric quad, hybrid, n modes	7	44	CAXA8Hn
CAXA8Rn	8 Node quadratic asymmetric quad, reduced, n modes	7	44	CAXA8Rn
CAXA8RHn	8 Node quadratic asymmetric quad, reduced, hybrid, n modes	7	44	CAXA8RHn
CGAX3	3 Node generalized linear axisymmetric triangle	3	43	CGAX3
CGAX3H	3 Node generalized linear axisymmetric triangle, hybrid	3	43	CGAX3H
CGAX4	4 Node generalized linear axisymmetric quad	6	43	CGAX4
CGAX4H	4 Node generalized linear axisymmetric quad, hybrid	6	43	CGAX4H
CGAX4R	4 Node generalized linear axisymmetric quad, reduced	6	43	CGAX4R
CGAX4RH	4 Node generalized linear axisymmetric quad, reduced, hybrid	6	43	CGAX4RH
CGAX6	6 Node generalized quadratic axisymmetric triangle	4	43	CGAX6
CGAX6H	6 Node generalized quadratic axisymmetric triangle, hybrid	4	43	CGAX6H
CGAX8	8 Node generalized quadratic axisymmetric quad	7	43	CGAX8
CGAX8H	8 Node generalized quadratic axisymmetric quad, hybrid	7	43	CGAX8H
CGAX8R	8 Node generalized quadratic axisymmetric quad, reduced	7	43	CGAX8R
CGAX8RH	8 Node generalized quadratic axisymmetric quad, reduced, hybrid	7	43	CGAX8RH
CPE3	3 Node linear plane strain triangle	3	41	CPE3
CPE3H	3 Node linear plane strain triangle, hybrid	3	41	CPE3H
CPE4	4 Node linear plane strain quad	6	41	CPE4
CPE4H	4 Node linear plane strain quad, hybrid	6	41	CPE4H
CPE4R	4 Node linear plane strain quad, reduced	6	41	CPE4R
CPE4RH	4 Node linear plane strain quad, reduced, hybrid	6	41	CPE4RH
CPE6	6 Node quadratic plane strain triangle	4	41	CPE6
CPE6H	6 Node quadratic plane strain triangle, hybrid	4	41	CPE6H
CPE8	8 Node quadratic plane strain quad	7	41	CPE8
CPE8H	8 Node quadratic plane strain quad, hybrid	7	41	CPE8H
CPE8R	8 Node quadratic plane strain quad, reduced	7	41	CPE8R
CPE8RH	8 Node quadratic plane strain quad, reduced, hybrid	7	41	CPE8RH
CPS3	3 Node linear plane stress triangle	3	40	CPS3
CPS4	4 Node linear plane stress quad	6	40	CPS4
CPS6	6 Node quadratic plane stress triangle	4	40	CPS6
CPS8	6 Node quadratic plane stress quad	7	40	CPS8
DASHPOT1	1 Node to ground dashpot	0	2	DASHPOT1
DASHPOTA	2 Node dashpot	1	22	DASHPOTA
DC1D2	2 Node heat transfer link	1	29	DC1D2
DC1D3	3 Node heat transfer link	2	29	DC1D3
DC2D3	3 Node linear heat transfer triangle	3	48	DC2D3
DC2D4	4 Node linear heat transfer quad	6	48	DC2D4
DC2D6	6 Node quadratic heat transfer triangle	4	48	DC2D6
DC2D8	8 Node quadratic heat transfer quad	7	48	DC2D8
DC3D4	4 Node linear heat transfer tetrahedron	9	61	DC3D4
DC3D6	6 Node linear heat transfer wedge	12	61	DC3D6
DC3D8	8 Node linear heat transfer brick	15	61	DC3D8
DC3D10	10 Node quadratic heat transfer tetrahedron	10	61	DC3D10
DC3D15	15 Node quadratic heat transfer wedge	13	61	DC3D15
DC3D20	20 Node quadratic heat transfer brick	16	61	DC3D20
DCAX3	3 Node linear axisymmetric heat transfer triangle	3	49	DCAX3
DCAX4	4 Node linear axisymmetric heat transfer quad	6	49	DCAX4
DCAX6	6 Node quadratic axisymmetric heat transfer triangle	4	49	DCAX6

DCAX8	8 Node quadratic axisymmetric heat transfer quad	7	49	DCAX8
DINTER1	1 Node per side heat transfer interface	30	85	DINTER1
DINTER2	2 Node per side heat transfer interface	31	85	DINTER2
DINTER2A	2 Node per side axisymmetric heat transfer interface	31	86	DINTER2A
DINTER3	3 Node per side heat transfer interface	32	85	DINTER3
DINTER3A	3 Node per side axisymmetric heat transfer interface	32	86	DINTER3A
DINTER4	4 Node per face heat transfer interface	34	85	DINTER4
DINTER8	8 Node per face heat transfer interface	35	85	DINTER8
DSAX1	2 Node axisymmetric heat transfer shell	1	30	DSAX1
DSAX2	3 Node axisymmetric heat transfer shell	2	30	DSAX2
GAPUNI	2 Node gap	30	83	GAPUNI
INTER2	2 Node per side interface	31	83	INTER2
INTER2A	2 Node per side axisymmetric interface	31	84	INTER2A
INTER3	3 Node per side interface	32	83	INTER3
INTER3A	3 Node per side axisymmetric interface	32	84	INTER3A
INTER4	4 Node per face interface	34	83	INTER4
INTER8	8 Node per face interface	35	83	INTER8
INTER9	9 Node per face interface	36	83	INTER9
IRS21A	3 Node axisymmetric rigid surface	41	28	IRS21A
ISL21A	3 Node axisymmetric slide line	1	28	ISL21A
M3D3	3 Node linear membrane triangle	3	45	M3D3
M3D4	4 Node linear membrane quad	6	45	M3D4
M3D4R	4 Node linear membrane quad, reduced	6	45	M3D4R
M3D6	6 Node quadratic membrane triangle	4	45	M3D6
M3D8	8 Node quadratic membrane quad	7	45	M3D8
M3D8R	8 Node quadratic membrane quad, reduced	7	45	M3D8R
MASS	1 Node point mass	0	3	MASS
S4R5	4 Node linear curved thin shell, reduced	6	46	S4R5
S8R5	8 Node quadratic curved thin shell, reduced	7	46	S8R5
SAX1	2 Node linear axisymmetric shell	1	27	SAX1
SAX2	3 Node quadratic axisymmetric shell	2	27	SAX2
SAXA1n	2 Node linear asymmetric shell, n modes	1	28	SAXA1n
SAXA2n	3 Node quadratic asymmetric shell, n modes	2	28	SAXA2n
SPRING1	1 Node to ground spring	0	0	SPRING1
SPRINGA	2 Node to node spring	1	20	SPRINGA
STRI35	3 Node linear curved thin shell	3	46	STRI35
STRI65	6 Node quadratic curved thin shell	4	46	STRI65
T2D2	2 Node linear 2D truss	1	23	T2D2
T2D2H	2 Node linear 2D truss, hybrid	1	23	T2D2H
T2D3	3 Node quadratic 2D truss	2	23	T2D3
T2D3H	3 Node quadratic 2D truss, hybrid	2	23	T2D3H
T3D2	2 Node linear 3D truss	1	24	T3D2
T3D2H	2 Node linear 3D truss, hybrid	1	24	T3D2H
T3D3	3 Node quadratic 3D truss	2	24	T3D3
T3D3H	3 Node quadratic 3D truss, hybrid	2	24	T3D3H

Return to Table of Contents

ANSYS

ANSYS is a finite element code primarily used for structural analysis. FEM Builder supports model definition, boundary condition creation, and specification of material properties. Interfaces for reading results files, reading coded database files, and writing coded database files are described in the following sections. This interface was written to support ANSYS 5.6.

ANSYS Read Results File Interface

This interface reads ANSYS file output files. Nodes, elements, coordinate systems, and analysis results are extracted from the file data. The following analysis results are supported.

Nodal Results

Nodal Results	Displacements
	Velocity
	Pressure
	Temperature

Element Nodal Results

Element Nodal Results	Stress
	Strain – Elastic
	Strain – Plastic
	Strain – Creep
	Strain Energy Density
	Temperature

Top

ANSYS Read Coded Database File Interface

This interface reads ANSYS coded database files written by the ANSYS CDWRITE command and only supports the coded database commands listed in the ANSYS documentation. Nodes, elements, coordinate systems, loads, boundary conditions, and material properties are extracted from the file data.

Nodes

The interface reads node label, coordinates, and displacement coordinate system.

Elements

The Fem Builder supported element types (geometry and analysis types) are found in Appendix - Element Library: Table 1 and Appendix - Element Library: Table 2 respectively. Table Appendix - ABAQUS Support: Table 1 lists each supported ANSYS element type and the geometry type and analysis type that it will be assigned as it is read into Fem Builder.

Coordinate Systems

The interface reads a coordinate system's origin, orientation, and type (e.g. rectangular, cylindrical, spherical).

Loads and Boundary Conditions

For structural and thermal results, the following table lists ANSYS supported loads and their supported counterparts in Fem Builder.

Type	ANSYS	Fem Builder
Structural	Translations	Restraints (1-3)

	Rotations	Restraints (4-6)
	Forces	Nodal Forces (1-3)
	Moments	Nodal Forces (4-6)
	Pressure	Pressure
	Temperature	Temperature
	Fluence	Not supported
Thermal	Temperature	Fixed Temperature
	Heat Flow Rate	Not supported
	Convection	Convection
	Heat Flux	Heat Flux
	Infinite Surface	Not supported
	Heat Generation Rate	Heat Source (on an element or node)

Material Properties

The following table lists the ANSYS supported material properties and their supported counterparts in Fem Builder.

ANSYS	Fem Builder
Elastic Modulus Poisson's Ratio Shear Modulus	Elastic Modulus – Isotropic Elastic Modulus - Orthotropic Elastic Modulus – Anisotropic
Thermal Expansion Coef.	Expansion – Isotropic Expansion – Orthotropic Expansion – Anisotropic
Thermal Conductivity	Conductivity – Isotropic Conductivity – Orthotropic Conductivity – Anisotropic
Mass Density	Density
Specific Heat	Specific Heat – CP
Emissivity	Emissivity

Top

ANSYS Write Coded Database File Interface

This interface writes ANSYS coded database files and only supports the coded database commands listed in the ANSYS documentation. Nodes, elements, coordinate systems, loads, boundary conditions, and material properties are extracted from Fem Builder and written out in an ANSYS coded database file.

Nodes

The interface writes node label, coordinates, and rotation angles.

Elements

The Fem Builder supported element types (geometry and analysis types) are found in Appendix - Element Library: Table 1 and Appendix - Element Library: Table 2 respectively. Tables Appendix - Ansys Support: Tables 2 to 7 lists each Fem Builder geometry and analysis type and the supported ANSYS element type that it will be assigned as the ANSYS coded database file is written. Some Fem Builder element types, such as wedges, tetrahedrons, and triangles, are supported in ANSYS by writing it as a degenerate form of a supported element type (e.g. a linear triangle would be written as a degenerate form of ANSYS element type PLANE42).

Coordinate Systems

The interface writes a coordinate system's origin, rotation angles, and type (e.g. rectangular, cylindrical, spherical).

Loads and Boundary Conditions

For structural and thermal results, the following table lists ANSYS supported loads and their supported counterparts in Fem Builder.

Type	ANSYS	Fem Builder
Structural	Translations	Restraints (1-3)
	Rotations	Restraints (4-6)
	Forces	Nodal Forces (1-3)
	Moments	Nodal Forces (4-6)
	Pressure	Pressure
	Temperature	Temperature
	Fluence	Not supported
Thermal	Temperature	Fixed Temperature
	Heat Flow Rate	Not supported
	Convection	Convection
	Heat Flux	Heat Flux
	Infinite Surface	Not supported
	Heat Generation Rate	Heat Source (on an element or node)

Material Properties

The following table lists the ANSYS supported material properties and their supported counterparts in Fem Builder.

ANSYS	Fem Builder
Elastic Modulus Poisson's Ratio Shear Modulus	Elastic Modulus – Isotropic Elastic Modulus - Orthotropic Elastic Modulus – Anisotropic
Thermal Expansion Coef.	Expansion – Isotropic Expansion – Orthotropic Expansion – Anisotropic
Thermal Conductivity	Conductivity – Isotropic Conductivity – Orthotropic Conductivity – Anisotropic
Mass Density	Density
Specific Heat	Specific Heat – CP
Emissivity	Emissivity

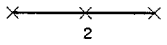
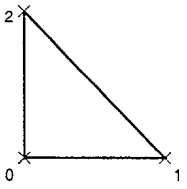
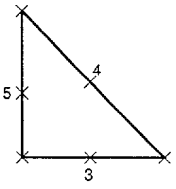
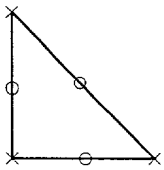
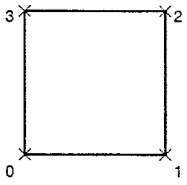
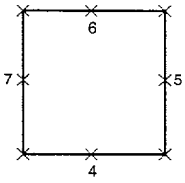
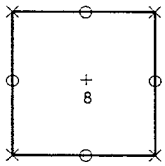
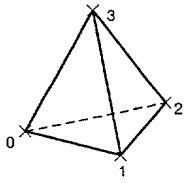
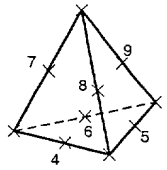
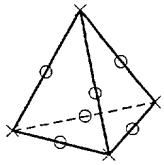
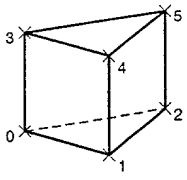
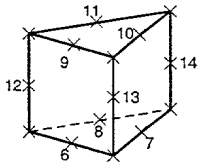
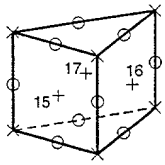
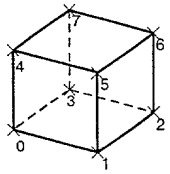
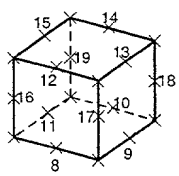
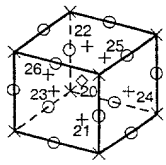
Top

Return to Table of Contents

Appendix - Element Library

Node Key:

- × Corner and Mid-side node (Required)
- ○ Mid-side node (Optional)
- \+ Face center node (Optional)
- ◇ Element center node (Optional)

TYPE	LINEAR	PARABOLIC	VARIABLE
Point			
Line			
Triangle Face 0=0,1, 3 Face 1=1,2, 4 Face 2=2,0, 5			
Quadrilateral Face 0=0,1, 4 Face 1=1,2, 5 Face 2=2,3, 6 Face 3=3,0, 7			
Tetrahedron Face 0=0,2,1, 6,5,4 Face 1=0,1,3, 4,8,7 Face 2=1,2,3, 5,9,8 Face 3=2,0,3, 6,7,9			
Wedge Face 0=0,2,1, 8,7,6 Face 1=3,4,5, 9,10,11 Face 2=0,1,4,3, 6,13,9,12,15 Face 3=1,2,5,4, 7,14,10,13,16 Face 4=2,0,3,5, 8,12,11,14,17			
Brick Face 0=0,3,2,1, 11,10,9,8,21 Face 1=4,5,6,7, 12,13,14,15,22 Face 2=0,1,5,4, 8,17,12,16,23 Face 3=1,2,6,5, 9,18,13,17,24 Face 4=2,3,7,6, 10,19,14,18,25 Face 5=3,0,4,7, 11,16,15,19,26			

Element Type Support

A geometry type, such as linear triangle or linear quadrilateral, and an analysis type, such as plane stress or plane strain, define a Fem Builder element types. Table A.1-1 lists Fem Builder geometry types. Table A.1-2 lists Fem Builder analysis types.

Table 1: Fem Builder geometry types

0	Point
1	Linear Line
2	Quadratic Line
3	Linear Triangle
4	Quadratic Triangle
5	Variable Triangle
6	Linear Quad
7	Quadratic Quad
8	Variable Quad
9	Linear Tetra
10	Quadratic Tetra
11	Variable Tetra
12	Linear Wedge
13	Quadratic Wedge
14	Variable Wedge
15	Linear Brick
16	Quadratic Brick
17	Variable Brick
30	2 Node Interface: Opposing sides, point
31	4 Node Interface: Opposing sides, line
32	6 Node Interface: Opposing sides, line
33	6 Node Interface: Opposing faces
34	8 Node Interface: Opposing faces
35	16 Node Interface: Opposing faces
36	18 Node Interface: Opposing faces
40	1 Node + ref node Interface: Rigid, point
41	2 Node + ref node Rigid interface, line
42	3 Node + ref node Rigid interface, line
43	3 Node + ref node Rigid interface
44	4 Node + ref node Rigid interface
45	8 Node + ref node Rigid interface
46	9 Node + ref node Rigid interface
50	2D Infinite, 2-9 Node
51	3D Infinite, 4-27 Node
60	N Node User defined

Table 2: Fem Builder Analysis Types

Types for point elements

0	Node to ground translational spring
1	Node to ground rotational spring
2	Node to ground damper
3	Point mass

Types for line elements

20	Node to node translational spring
21	Node to node rotational spring
22	Node to node damper
23	2D Truss
24	3D Truss
25	2D Beam
26	3D Beam
27	Axisymmetric shell
28	Asymmetric shell
29	Heat transfer
30	Axisymmetric heat transfer
31	Slide line

Types for planar elements

40	Plane stress
41	Plane strain
42	Generalized plane strain
43	Axisymmetric
44	Asymmetric
45	Membrane
46	Shell
47	Fluid
48	Heat transfer
49	Axisymmetric heat transfer

Types for 3D elements

60	Structural
61	Heat transfer
62	Fluid

Types for interface elements

80	1 DOF structural interface
81	2 DOF structural interface
82	3 DOF structural interface
83	Structural interface
84	Axisymmetric structural interface
85	Heat transfer interface
86	Axisymmetric heat transfer interface

Types for infinite elements

100	Plane stress
101	Plane strain
102	Axisymmetric
103	Solid

Type for user defined elements

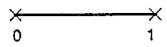
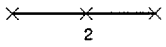
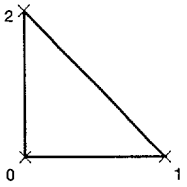
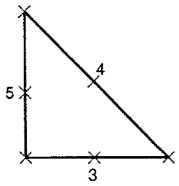
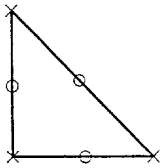
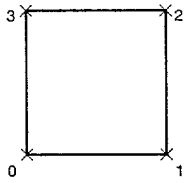
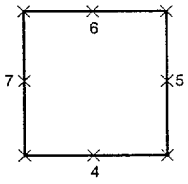
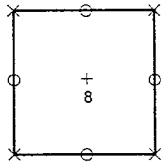
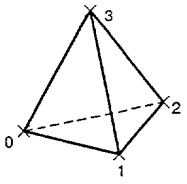
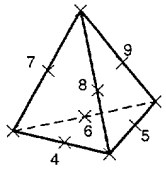
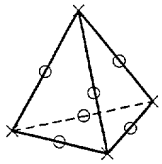
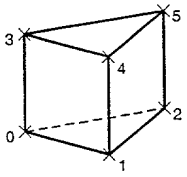
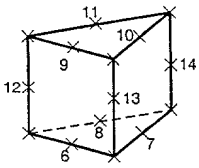
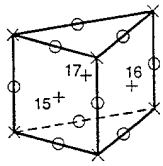
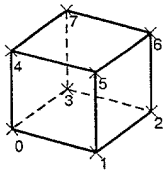
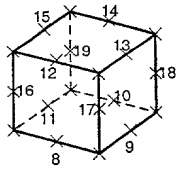
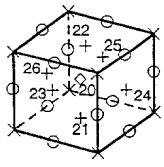
120	User defined

Return to Table of Contents

Appendix - Element Library

Node Key:

- × Corner and Mid-side node (Required)
- ○ Mid-side node (Optional)
- \+ Face center node (Optional)
- ◇ Element center node (Optional)

TYPE	LINEAR	PARABOLIC	VARIABLE
Point			
Line			
Triangle Face 0=0,1, 3 Face 1=1,2, 4 Face 2=2,0, 5			
Quadrilateral Face 0=0,1, 4 Face 1=1,2, 5 Face 2=2,3, 6 Face 3=3,0, 7			
Tetrahedron Face 0=0,2,1, 6,5,4 Face 1=0,1,3, 4,8,7 Face 2=1,2,3, 5,9,8 Face 3=2,0,3, 6,7,9			
Wedge Face 0=0,2,1, 8,7,6 Face 1=3,4,5, 9,10,11 Face 2=0,1,4,3, 6,13,9,12,15 Face 3=1,2,5,4, 7,14,10,13,16 Face 4=2,0,3,5, 8,12,11,14,17			
Brick Face 0=0,3,2,1, 11,10,9,8,21 Face 1=4,5,6,7, 12,13,14,15,22 Face 2=0,1,5,4, 8,17,12,16,23 Face 3=1,2,6,5, 9,18,13,17,24 Face 4=2,3,7,6, 10,19,14,18,25 Face 5=3,0,4,7, 11,16,15,19,26			

Element Type Support

A geometry type, such as linear triangle or linear quadrilateral, and an analysis type, such as plane stress or plane strain, define a Fem Builder element types. Table A.1-1 lists Fem Builder geometry types. Table A.1-2 lists Fem Builder analysis types.

Table 1: Fem Builder geometry types

0	Point
1	Linear Line
2	Quadratic Line
3	Linear Triangle
4	Quadratic Triangle
5	Variable Triangle
6	Linear Quad
7	Quadratic Quad
8	Variable Quad
9	Linear Tetra
10	Quadratic Tetra
11	Variable Tetra
12	Linear Wedge
13	Quadratic Wedge
14	Variable Wedge
15	Linear Brick
16	Quadratic Brick
17	Variable Brick
30	2 Node Interface: Opposing sides, point
31	4 Node Interface: Opposing sides, line
32	6 Node Interface: Opposing sides, line
33	6 Node Interface: Opposing faces
34	8 Node Interface: Opposing faces
35	16 Node Interface: Opposing faces
36	18 Node Interface: Opposing faces
40	1 Node + ref node Interface: Rigid, point
41	2 Node + ref node Rigid interface, line
42	3 Node + ref node Rigid interface, line
43	3 Node + ref node Rigid interface
44	4 Node + ref node Rigid interface
45	8 Node + ref node Rigid interface
46	9 Node + ref node Rigid interface
50	2D Infinite, 2-9 Node
51	3D Infinite, 4-27 Node
60	N Node User defined

Table 2: Fem Builder Analysis Types

Types for point elements

0	Node to ground translational spring
1	Node to ground rotational spring
2	Node to ground damper
3	Point mass

Types for line elements

20	Node to node translational spring
21	Node to node rotational spring
22	Node to node damper
23	2D Truss
24	3D Truss
25	2D Beam
26	3D Beam
27	Axisymmetric shell
28	Asymmetric shell
29	Heat transfer
30	Axisymmetric heat transfer
31	Slide line

Types for planar elements

40	Plane stress
41	Plane strain
42	Generalized plane strain
43	Axisymmetric
44	Asymmetric
45	Membrane
46	Shell
47	Fluid
48	Heat transfer
49	Axisymmetric heat transfer

Types for 3D elements

60	Structural
61	Heat transfer
62	Fluid

Types for interface elements

80	1 DOF structural interface
81	2 DOF structural interface
82	3 DOF structural interface
83	Structural interface
84	Axisymmetric structural interface
85	Heat transfer interface
86	Axisymmetric heat transfer interface

Types for infinite elements

100	Plane stress
101	Plane strain
102	Axisymmetric
103	Solid

Type for user defined elements

120	User defined

Return to Table of Contents

Appendix - ANSYS Element Type Support

Table 1 lists each supported ANSYS element type and the geometry type and analysis type that it is assigned when reading into Fem Builder. It also lists the ANSYS element types that are written to ANSYS from Fem Builder. ANSYS element types not directly supported on output will be mapped to the element type listed in the ANSYS output column.

Table 1: ANSYS element type support

ANSYS Input		FEM Builder Type		ANSYS Output
Name	***Description***	***Geometry***	***Analysis***	***Map to Element***
LINK1	2D Spar	1	23	1
*PLANE2	2D 6 node Triangle, Plane Stress	4	40	82
	2D 6 node Triangle, Plane Strain		41	82,84,108
	2D 6 node Triangle, Axisymmetric		43	82,84,108
BEAM3	2D Elastic Beam	1	25	3
BEAM4	3D Elastic Beam	1	26	4
*SOLID5	3D 8 node Coupled Solid (brick or wedge)	12,15	60	45,64,86,107
			61	70
COMBIN7	Revolute Joint	not supported		
LINK8	3D Spar	1	24	8
INFIN9	2D Infinite Boundary	not supported		
*LINK10	3D Spar - Tension only	1	23	1
*LINK11	3D Actuator	1	23	1
CONTAC12	2D Point to Point Contact	30	81	12
PLANE13	2D Coupled Field (linear quad,tri)	3,6	40	42
			41	42,56,106,182
			43	42,56,106,182
			48	55,57
			49	55,75
COMBIN14	Spring and/or Damper (1D,2D, and 3D)	1	20	14
			21	14
			22	14
PIPE16	Elastic Straight Pipe	1	26	4
PIPE17	Elastic Pipe Tee	not supported		
PIPE18	Elastic Curved Pipe (elbow)	not supported		
SURF19	2D Surface Effect	not supported		
PIPE20	Plastic Straight Pipe	not supported		
MASS21	Structural Point Mass	0	3	21,71
SURF22	3D Surface Effect	not supported		
*BEAM23	2D Linear Beam	1	25	3
*BEAM24	3D Linear Beam	1	26	4
PLANE25	Axisymmetric with Asymetric loads	3,6	44	25
CONTAC26	2D Point to Ground Contact	40	81	26
MATRIX27	Stiffness, Damping, or Mass Matrix	not supported		
*SHELL28	Shear/Twist Panel	6	46	63,181
FLUID29	2D Acoustic Fluid	not supported		
FLUID30	3D Acoustic Fluid	not supported		
*LINK31	Radiation (Is this a boundary condition?)	1	29,30	32
			29	33

LINK32	2D Conduction	1	29,30	32
LINK33	3D Conduction	1	29	33
Name	***Description***	***Geometry***	***Analysis***	***Map to Element***
*LINK34	Convection (Is this a boundary condition?)	1	29,30	32
			29	33
*PLANE35	2D 6 node Triangular Thermal Solid	4	48,49	77
SOURC36	Current Source	not supported		
COMBIN37	Control	not supported		
FLUID38	Dynamic Fluid Coupling	not supported		
*COMBIN39	Nonlinear Spring (translation)	1	20	14
	(rotation)		21	14
COMBIN40	Combination	30	80,85	40
SHELL41	Membrane Shell	3,6	45	41
PLANE42	2D 4 node Structural Solid (stress)	3,6	40	42,182
	(strain)		41	42,182
	(axisymmetric)		43	42,182
*SHELL43	Plastic Large Strain Shell	3,6	46	63,181
BEAM44	3D Offset Tapered Unsymmetric Beam	not supported		
SOLID45	3D Structural Solid	9,12,15	60	45
*SOLID46	3D Layered Structural Solid	9,12,15	60	45,64,86,107
INFIN47	3D Infinite Boundary	not supported		
CONTAC48	2D Point to Surface Contact	not supported		
CONTAC49	3D Point to Surface Contact	not supported		
MATRIX50	Superelement	not supported		
*SHELL51	Axisymmetric Structual Shell	1	27	61
CONTAC52	3D Point to Point Contact	30	82,83	52
PLANE53	Magnetic	not supported		
*BEAM54	2D Offset Tapered Unsymmetric Beam	not supported		3
PLANE55	2D Thermal Solid	3,6	48	55,57
			49	55,75
HYPER56	2D 4 node Hyperelastic Solid	3,6	41,43	42,56,106,182
SHELL57	3D 4 node Thermal Shell	3,6	48	57
*HYPER58	3D 8 node Hyperelastic Solid	9,12,15	60	45,64,86,107
PIPE59	Immersed Pipe or Cable	not supported		
PIPE60	Plastic Curved Pipe (elbow)	not supported		
SHELL61	2 node Axisymmetric with Asymetric Loads	1	27	61
SOLID62	Magnetic Solid	not supported		
*SHELL63	3D 4 node Elastic Shell	3,6	46	63,181
SOLID64	3D 8 node Anisotropic Solid	9,12,15	60	45,64,86,107
*SOLID65	Reinforced Concrete	9,12,15	60	45,64,86,107
*FLUID66	3D Thermal Fluid Pipe	1	29	32,33
*PLANE67	4 node Thermal Electric Solid, Linear	3,6	48	55,57
*LINK68	Thermal Electric Line	1	29	32,33
*SOLID69	3D Thermal Electric Solid	9,12,15	61	70
SOLID70	3D Thermal Solid	9,12,15	61	70
MASS71	Thermal Point Mass	0	3	21,71
*SOLID72	3D 4 node Tetrahedron with Rotations	9	60	72,86,107
*SOLID73	3D 8 node Structural Brick with Rotations	9,12,15	60	45,64,86,107
*HYPER74	2D 8 node Hyper-elastic Solid	4,7	41,43	82,84,108
PLANE75	4 node Thermal Axisymmetric with Asymmetric Loads	3,6	49	55,75
PLANE77	2D 8 node Thermal Solid	4,7	48	77
			49	77,78
PLANE78	8 node Thermal Axisymmetric with Asymmetric	4,7	49	77,78

	Loads			
FLUID79	2D Contained Fluid	not supported		
Name	***Description***	***Geometry***	***Analysis***	***Map to Element***
FLUID80	3D Contained Fluid	not supported		
FLUID81	Axisymmetric-Harmonic Contained Fluid	not supported		
PLANE82	2D 8 node Structural Solid	4,7	40	82
			41	82,84,108
			43	82,84,108
*PLANE83	8 node Axisymmetric with Asymmetric Loads	4,7	43	82,84,108
HYPER84	2D 8 node Hyperelastic Structural Solid	4,7	43	82,84,108
HYPER86	3D 8 node Hyperelastic Structural Solid	9,12,15	60	45,64,86,107
*SOLID87	10 node Thermal Tetrahedron	10	61	90
*VISCO88	2D 8 node Viscoelastic Solid	4,7	41,43	82,84,108
*VISCO89	3D 20 node Viscoelastic Solid	9,12,15	60	45,64,86,107
SOLID90	20 node Thermal Solid	10,13,16	61	90
*SHELL91	8 node Nonlinear Layered Structural Shell	4,7	46	93,99
*SOLID92	10 node Structural Tetrahedron	10	60	92,158
SHELL93	8 node Structural Shell	4,7	46	93,99
SOLID95	20 node Structural Solid	10,13,16	60	95,185
SOLID96	3D Magnetic Scalar Solid	not supported		
SOLID97	3D Magnetic Solid	not supported		
*SOLID98	10 node Coupled Field Tetrahedron	10	60	95,185
			61	90
SHELL99	8 node Layered Shell	4,7	46	93,99
VISCO106	2D 4 node Large Strain (viscoelastic)	3,6	41,43	42,56,106,182
VISCO107	3D 8 node Large Strain (viscoelastic)	9,12,15	60	45,64,86,107
VISCO108	2D 8 node Large Strain (viscoelastic)	4,7	41,43	82,84,108
INFIN110	2D 4 node Infinite Solid	50	100,101, axi	110
		51	100,101, axi	110
INFIN111	3D 8 node Infinite Solid	51	102	111
		not supported		
INTER115	3D Magnetic Interface	not supported		
*FLUID116	3D 2 node Thermal Fluid Pipe	1	29	32,33
SOLID117	3D Magnetic Solid	not supported		
HF119	3D Tetrahedral High-frequency	not supported		
HF120	3D Brick/Wedge High-frequency	not supported		
PLANE121	2D 8 node Electrostatic Solid	not supported		
SOLID122	3D 20 node Electrostatic Solid	not supported		
SOLID123	3D 10 node Tetrahedral Electrostatic Solid	not supported		
CIRCU124	General Circuit	not supported		
FLUID129	2D Infinite Acoustic	not supported		
FLUID130	3D Infinite Acoustic	not supported		
*FLUID141	2D 4 node Thermal Fluid	3,6	48	55,57
FLUID142	3D 8 node Thermal Fluid	9,12,15	62	142
*SHELL143	3D 4 node Plastic Shell	3,6	46	63,181
*PLANE145	2D 8 node Structural P-element	4,7	40,41,43	82,84,108
*PLANE146	2D 6 node Structual P-element	4	40,41,43	82,84,108
*SOLID147	3D 20 node Structural Brick P-element	10,13,16	60	95,185
*SOLID148	3D 10 node Structrual Tetrahedron	10	60	95,185
*SHELL150	3D 8 node Structural Shell P-element	4,7	46	93,99
SURF151	2D Thermal Surface Effect	not supported		
SURF152	3D Thermal Surface Effect	not supported		

SURF153	2D Structural Surface Effect	not supported		
SURF154	3D Structural Surface Effect	not supported		
Name	***Description***	***Geometry***	***Analysis***	***Map to Element***
*SHELL157	4 node Thermal-electric Shell	3,6	48	55,57
*HYPER158	3D 10 node Hyperelastic Tetrahedron	10	60	158
LINK160	3D Explicit Spar	not supported		
BEAM161	3D Explicit Beam	not supported		
SHELL163	Explicit Thin Structural Shell	not supported		
SOLID164	3D Explicit Structural Shell	not supported		
COMBIN165	Explicit Spring-Damper	not supported		
MASS166	3D Explicit Structural Mass	not supported		
LINK167	Explicit Spar (tension only)	not supported		
TARGE169	2D target segmet (slide line)	supported, not as an element		
TARGE170	3D Target Surface	not supported		
CONTA171	2D Surface to Surface Contact	supported, not as an element		
CONTA172	2D 3 node Surface to Surface Contact	supported, not as an element		
CONTA173	3D Surface to Surface Contact	supported, not as an element		
CONTA174	3D 8 node Surface to Surface Contact	supported, not as an element		
SHELL181	3D 4 Node Structural Shell (reduced integrate)	3,6	46	63,181
PLANE182	2D 4 node Structural Solid (reduced integrate)	3,6	40	42,182
			41	42,56,106,182
			43	42,56,106,182
SOLID185	3D 8 node Structural Solid (reduced integrate)	10,13,16	60	95,185
BEAM188	3D 2 node Finite Strain Beam	not supported		
BEAM189	3D 3 node Finite Strain Beam	not supported		

* Element type not supported on output to ANSYS. Output to ANSYS uses the mapped element type.

A.2 Element mapping from Fem Builder to ANSYS

The following tables map the Fem Builder geometry and analysis types to the supported ANSYS elements that are written in the Fem Builder to ANSYS output file.

Table 2: Types for point elements

	Geometry type	Point
Analysis type		0
Node to ground translational spring	0	
Node to ground rotational spring	1	
Node to ground damper	2	
Point mass	3	21,71

Table 3: Types for line elements

Analysis type	Geometry type	Linear Line	Quadratic Line
		1	2
Node to node translational spring	20	14	
Node to node rotational spring	21	14	
Node to node damper	22	14	
2D Truss	23	1	1
3D Truss	24	8	8
2D Beam	25	3	3
3D Beam	26	4	189
Axisymmetric shell	27	61	61
Asymmetric shell	28	61	61
Heat transfer	29	32,33	32,33
Axisymmetric heat transfer	30	32	32

Table 4: Types for planar elements

Analysis type	Geometry type	Linear Triangle	Quadratic Triangle	Variable Triangle	Linear Quad	Quadratic Quad	Variable Quad
		3	4	5	6	7	8
Plane stress	40	42,182	82	82	42,182	82	82
Plane strain	41	42,56,106,182	82,84,108	82,84,108	42,56,106,182	82,84,108	82,84,108
Generalized plane strain	42	42,56,106,182	82,84,108	82,84,108	42,56,106,182	82,84,108	82,84,108
Axisymmetric	43	42,56,106,182	82,84,108	82,84,108	42,56,106,182	82,84,108	82,84,108
Asymmetric	44	25	83,84	83,84	25	83,84	83,84
Membrane	45	41			41		
Shell	46	63,181	93,99	93,99	63,181	93,99	93,99
Fluid	47	141	141	141	141	141	141
Heat transfer	48	55,57	77	77	55,57	77	77
Axisymmetric heat transfer	49	55,75	77,78	77,78	55,75	77,78	77,78

Table 5: Types for 3 dimensional elements

Geometry type	Linear Tetra	Quadratic Tetra	Variable Tetra	Linear Wedge	Quadratic Wedge	Variable Wedge	Linear Brick	Quadratic Brick	Variable Brick
Analysis type	9	10	11	12	13	14	15	16	17
Structural 60	72,86,107	92,158	92,158	45,64,86,107	95,185	95,185		95,185	95,185
Heat transfer 61	70	90	90	70	90	90	70	90	90
Fluid 62	142	142	142				142	142	142

Table 6: Types for interface elements

Geometry Type	2 Node Interface: Opposing sides	4 Node Interface: Opposing sides	6 Node Interface: Opposing sides	6 Node Interface: Opposing faces	8 Node Interface: Opposing faces	16 Node Interface: Opposing faces	18 Node Interface: Opposing faces	1 Node + ref node Interface: Rigid	2 Node + ref node Rigid interface	3 Node + ref node Rigid interface	3 Node + ref node Rigid interface	4 Node + ref node Rigid interface	8 Node + ref node Rigid interface	9 Node + ref node Rigid interface
Analysis Type	30	31	32	33	34	35	36	40	41	42	43	44	45	46
1 DOF structural interface 80	40													
2 DOF structural interface 81	12							26						
3 DOF structural interface 82	52													
Structural interface 83	52													
Axisymmetric structural interface 84														
Heat transfer interface 85	40													
Axisymmetric heat transfer interface 86														

Table 7: Types for infinite elements

Geometry type	2D Infinite, 2-9 Node	3D Infinite, 4-27 Node	N Node User defined
Analysis type	50	51	60
Plane stress 100	110		
Plane strain 101	110		
Axisymmetric 102	110		

Solid	103		111	
Type for user defined elements				
User defined	120			

Return to Table of Contents

Appendix - ANSYS Element Type Support

Table 1 lists each supported ANSYS element type and the geometry type and analysis type that it is assigned when reading into Fem Builder. It also lists the ANSYS element types that are written to ANSYS from Fem Builder. ANSYS element types not directly supported on output will be mapped to the element type listed in the ANSYS output column.

Table 1: ANSYS element type support

ANSYS Input		FEM Builder Type		ANSYS Output
Name	***Description***	***Geometry***	***Analysis***	***Map to Element***
LINK1	2D Spar	1	23	1
*PLANE2	2D 6 node Triangle, Plane Stress	4	40	82
	2D 6 node Triangle, Plane Strain		41	82,84,108
	2D 6 node Triangle, Axisymmetric		43	82,84,108
BEAM3	2D Elastic Beam	1	25	3
BEAM4	3D Elastic Beam	1	26	4
*SOLID5	3D 8 node Coupled Solid (brick or wedge)	12,15	60	45,64,86,107
			61	70
COMBIN7	Revolute Joint	not supported		
LINK8	3D Spar	1	24	8
INFIN9	2D Infinite Boundary	not supported		
*LINK10	3D Spar - Tension only	1	23	1
*LINK11	3D Actuator	1	23	1
CONTAC12	2D Point to Point Contact	30	81	12
PLANE13	2D Coupled Field (linear quad,tri)	3,6	40	42
			41	42,56,106,182
			43	42,56,106,182
			48	55,57
			49	55,75
COMBIN14	Spring and/or Damper (1D,2D, and 3D)	1	20	14
			21	14
			22	14
PIPE16	Elastic Straight Pipe	1	26	4
PIPE17	Elastic Pipe Tee	not supported		
PIPE18	Elastic Curved Pipe (elbow)	not supported		
SURF19	2D Surface Effect	not supported		
PIPE20	Plastic Straight Pipe	not supported		
MASS21	Structural Point Mass	0	3	21,71
SURF22	3D Surface Effect	not supported		
*BEAM23	2D Linear Beam	1	25	3
*BEAM24	3D Linear Beam	1	26	4
PLANE25	Axisymmetric with Asymetric loads	3,6	44	25
CONTAC26	2D Point to Ground Contact	40	81	26
MATRIX27	Stiffness, Damping, or Mass Matrix	not supported		
*SHELL28	Shear/Twist Panel	6	46	63,181
FLUID29	2D Acoustic Fluid	not supported		
FLUID30	3D Acoustic Fluid	not supported		
*LINK31	Radiation (Is this a boundary condition?)	1	29,30	32
			29	33

LINK32	2D Conduction	1	29,30	32
LINK33	3D Conduction	1	29	33
Name	***Description***	***Geometry***	***Analysis***	***Map to Element***
*LINK34	Convection (Is this a boundary condition?)	1	29,30	32
			29	33
*PLANE35	2D 6 node Triangular Thermal Solid	4	48,49	77
SOURC36	Current Source	not supported		
COMBIN37	Control	not supported		
FLUID38	Dynamic Fluid Coupling	not supported		
*COMBIN39	Nonlinear Spring (translation)	1	20	14
	(rotation)		21	14
COMBIN40	Combination	30	80,85	40
SHELL41	Membrane Shell	3,6	45	41
PLANE42	2D 4 node Structural Solid (stress)	3,6	40	42,182
	(strain)		41	42,182
	(axisymmetric)		43	42,182
*SHELL43	Plastic Large Strain Shell	3,6	46	63,181
BEAM44	3D Offset Tapered Unsymmetric Beam	not supported		
SOLID45	3D Structural Solid	9,12,15	60	45
*SOLID46	3D Layered Structural Solid	9,12,15	60	45,64,86,107
INFIN47	3D Infinite Boundary	not supported		
CONTAC48	2D Point to Surface Contact	not supported		
CONTAC49	3D Point to Surface Contact	not supported		
MATRIX50	Superelement	not supported		
*SHELL51	Axisymmetric Structual Shell	1	27	61
CONTAC52	3D Point to Point Contact	30	82,83	52
PLANE53	Magnetic	not supported		
*BEAM54	2D Offset Tapered Unsymmetric Beam	not supported		3
PLANE55	2D Thermal Solid	3,6	48	55,57
			49	55,75
HYPER56	2D 4 node Hyperelastic Solid	3,6	41,43	42,56,106,182
SHELL57	3D 4 node Thermal Shell	3,6	48	57
*HYPER58	3D 8 node Hyperelastic Solid	9,12,15	60	45,64,86,107
PIPE59	Immersed Pipe or Cable	not supported		
PIPE60	Plastic Curved Pipe (elbow)	not supported		
SHELL61	2 node Axisymmetric with Asymetric Loads	1	27	61
SOLID62	Magnetic Solid	not supported		
*SHELL63	3D 4 node Elastic Shell	3,6	46	63,181
SOLID64	3D 8 node Anisotropic Solid	9,12,15	60	45,64,86,107
*SOLID65	Reinforced Concrete	9,12,15	60	45,64,86,107
*FLUID66	3D Thermal Fluid Pipe	1	29	32,33
*PLANE67	4 node Thermal Electric Solid, Linear	3,6	48	55,57
*LINK68	Thermal Electric Line	1	29	32,33
*SOLID69	3D Thermal Electric Solid	9,12,15	61	70
SOLID70	3D Thermal Solid	9,12,15	61	70
MASS71	Thermal Point Mass	0	3	21,71
*SOLID72	3D 4 node Tetrahedron with Rotations	9	60	72,86,107
*SOLID73	3D 8 node Structural Brick with Rotations	9,12,15	60	45,64,86,107
*HYPER74	2D 8 node Hyper-elastic Solid	4,7	41,43	82,84,108
PLANE75	4 node Thermal Axisymmetric with Asymmetric Loads	3,6	49	55,75
PLANE77	2D 8 node Thermal Solid	4,7	48	77
			49	77,78
PLANE78	8 node Thermal Axisymmetric with Asymmetric	4,7	49	77,78

SURF153	2D Structural Surface Effect	not supported		
SURF154	3D Structural Surface Effect	not supported		
Name	***Description***	***Geometry***	***Analysis***	***Map to Element***
*SHELL157	4 node Thermal-electric Shell	3,6	48	55,57
*HYPER158	3D 10 node Hyperelastic Tetrahedron	10	60	158
LINK160	3D Explicit Spar	not supported		
BEAM161	3D Explicit Beam	not supported		
SHELL163	Explicit Thin Structural Shell	not supported		
SOLID164	3D Explicit Structural Shell	not supported		
COMBIN165	Explicit Spring-Damper	not supported		
MASS166	3D Explicit Structural Mass	not supported		
LINK167	Explicit Spar (tension only)	not supported		
TARGE169	2D target segmet (slide line)	supported, not as an element		
TARGE170	3D Target Surface	not supported		
CONTA171	2D Surface to Surface Contact	supported, not as an element		
CONTA172	2D 3 node Surface to Surface Contact	supported, not as an element		
CONTA173	3D Surface to Surface Contact	supported, not as an element		
CONTA174	3D 8 node Surface to Surface Contact	supported, not as an element		
SHELL181	3D 4 Node Structural Shell (reduced integrate)	3,6	46	63,181
PLANE182	2D 4 node Structural Solid (reduced integrate)	3,6	40	42,182
			41	42,56,106,182
			43	42,56,106,182
SOLID185	3D 8 node Structural Solid (reduced integrate)	10,13,16	60	95,185
BEAM188	3D 2 node Finite Strain Beam	not supported		
BEAM189	3D 3 node Finite Strain Beam	not supported		

* Element type not supported on output to ANSYS. Output to ANSYS uses the mapped element type.

A.2 Element mapping from Fem Builder to ANSYS

The following tables map the Fem Builder geometry and analysis types to the supported ANSYS elements that are written in the Fem Builder to ANSYS output file.

Table 2: Types for point elements

	Geometry type	Point
Analysis type		0
Node to ground translational spring	0	
Node to ground rotational spring	1	
Node to ground damper	2	
Point mass	3	21,71

Table 3: Types for line elements

	Geometry type	Linear Line	Quadratic Line
Analysis type		1	2
Node to node translational spring	20	14	
Node to node rotational spring	21	14	
Node to node damper	22	14	
2D Truss	23	1	1
3D Truss	24	8	8
2D Beam	25	3	3
3D Beam	26	4	189
Axisymmetric shell	27	61	61
Asymmetric shell	28	61	61
Heat transfer	29	32,33	32,33
Axisymmetric heat transfer	30	32	32

Table 4: Types for planar elements

	Geometry type	Linear Triangle	Quadratic Triangle	Variable Triangle	Linear Quad	Quadratic Quad	Variable Quad
Analysis type		3	4	5	6	7	8
Plane stress	40	42,182	82	82	42,182	82	82
Plane strain	41	42,56,106,182	82,84,108	82,84,108	42,56,106,182	82,84,108	82,84,108
Generalized plane strain	42	42,56,106,182	82,84,108	82,84,108	42,56,106,182	82,84,108	82,84,108
Axisymmetric	43	42,56,106,182	82,84,108	82,84,108	42,56,106,182	82,84,108	82,84,108
Asymmetric	44	25	83,84	83,84	25	83,84	83,84
Membrane	45	41			41		
Shell	46	63,181	93,99	93,99	63,181	93,99	93,99
Fluid	47	141	141	141	141	141	141
Heat transfer	48	55,57	77	77	55,57	77	77
Axisymmetric heat transfer	49	55,75	77,78	77,78	55,75	77,78	77,78

Table 5: Types for 3 dimensional elements

Analysis type	Geometry type	Linear Tetra	Quadratic Tetra	Variable Tetra	Linear Wedge	Quadratic Wedge	Variable Wedge	Linear Brick	Quadratic Brick	Variable Brick
		9	10	11	12	13	14	15	16	17
Structural	60	72,86,107	92,158	92,158	45,64,86,107	95,185	95,185		95,185	95,185
Heat transfer	61	70	90	90	70	90	90	70	90	90
Fluid	62	142	142	142				142	142	142

Table 6: Types for interface elements

Analysis Type	Geometry Type	2 Node Interface: Opposing sides	4 Node Interface: Opposing sides	6 Node Interface: Opposing sides	6 Node Interface: Opposing faces	8 Node Interface: Opposing faces	16 Node Interface: Opposing faces	18 Node Interface: Opposing faces	1 Node + ref node Interface: Rigid	2 Node + ref node Rigid interface	3 Node + ref node Rigid interface	3 Node + ref node Rigid interface	4 Node + ref node Rigid interface	8 Node + ref node Rigid interface	9 Node + ref node Rigid interface
		30	31	32	33	34	35	36	40	41	42	43	44	45	46
1 DOF structural interface	80	40													
2 DOF structural interface	81	12							26						
3 DOF structural interface	82	52													
Structural interface	83	52													
Axisymmetric structural interface	84														
Heat transfer interface	85	40													
Axisymmetric heat transfer interface	86														

Table 7: Types for infinite elements

Analysis type	Geometry type	2D Infinite, 2-9 Node	3D Infinite, 4-27 Node	N Node User defined
		50	51	60
Plane stress	100	110		
Plane strain	101	110		
Axisymmetric	102	110		

Solid	103		111	
Type for user defined elements				
User defined	120			

Return to Table of Contents

Aschar Read Output File Interface

This interface reads Aschar *output (plot) files and extracts grid coordinates, material ids, centroidal temperatures of elements, coordinates of points on the ablation boundary, and temperatures at those points. This data is used to generate grids and temperature results.

The user has two grid options:

1. Save original grid and view the ablation profile as a function of time. Calculated nodal temperature and an ablation surface are saved for each time interval. Displacement results are created at each time step and displacements are defined to move nodes on the ablating surface of the original mesh to the current ablation surface location.
2. Use the ablation surface at a given time to edit and view the ablated structural grid. Nodal temperature results are calculated and saved at the time specified by the user.

The user also has the option of saving the centroidal temperatures of the elements.

Nodal temperature results along the ablation surface are interpolated from the given ablation surface centroidal results. The rest of the nodal temperature results are interpolated from the element centroidal temperature results given in the output file. The interpolation is done using a weighted inverse distance method. In other words, the smaller the distance between the centroid and the node the greater the influence it has in the interpolation of the nodal temperature.

*The Aschar output file is a binary file.

Return to Table of Contents

GridGen Read File Interface

This interface reads GridGen .grd and Plot 3D .p3d files and generates corresponding nodes and elements.

Return to Table of Contents

Ideas Master Series Read Universal File Interface

This interface reads Ideas Master Series (Ideas MS) universal files and extracts FEM data from supported data set sections. Supported data sets are as follows:

Name	Data Set
Header	151
Units	164
Material*	1710
Laminate Definitions	2415
Coordinate systems	2420
Nodes	2411
Elements	2412
Associated Element Data	748
Constraint Sets	754
Permanent Groups	2429
Restraint Sets	791
Load Sets	790
Amplitude	794, 796
Analysis Data	2414

Universal file data is converted from SI units into current model units as the file is read. After reading the universal file, FEM Builder converts axisymmetric models from the Ideas MS R-θ-Z convention to the FEM Builder 1-2-3 coordinate system convention.

* Only the material property name is extracted. Material properties are ignored.

Top

Ideas Master Series Write Universal File Interface

This interface writes Ideas Master Series universal files. Supported data sets are as follows:

Name	Data Set
Header	151
Units	164
Material	1710
Laminate Definitions	2415
Coordinate systems	2420
Nodes	2411
Elements	2412
Associated Element Data	748
Constraint Sets	754
Permanent Groups	2429
Restraint Sets	791
Load Sets	790
Analysis Data	2414

FEM Builder converts axisymmetric models from it's 1-2-3 coordinate system to the Ideas MS R-θ-Z convention.

Top
Return to Table of Contents

PATRAN Read Neutral File Interface

The neutral file is the way of transmitting PATRAN's database in and out of its system to other analysis programs. It is organized into "packets". Each packet contains the data for a fundamental unit of the model, such as the definition of a specific finite element. The following table shows the packets that are supported in Fem Builder's Read PATRAN interface.

Packet Number	Packet Description
25	File title
1	Node data
2	Element data
3	Material properties
5	Coordinate system
6	Distributed loads – pressures
7	Node forces
8	Node displacements
14	MPC data – constraints
21	Named components – groups
31	Grid data – points
32	Line data - curves

Nodes

The interface reads node label, coordinates, and displacement coordinate system.

Elements

The following table lists each supported PATRAN element geometry type and the corresponding Fem Builder element geometry type that supports it.

PATRAN Geometry Type	Number of nodes	FEM Builder Geometry Type
Bar	2	Linear Line
	3	Quadratic Line
Triangle	3	Linear Triangle
	6	Quadratic Triangle
	9	Not supported
Quad	4	Linear Quad
	8	Quadratic Quad
	12	Not supported
Tetra	4	Linear Tetra
	10	Quadratic Tetra
Wedge	6	Linear Wedge
	15	Quadratic Wedge
	24	Not supported
Hex	8	Linear Brick
	20	Quadratic Brick
	32	Not supported

The PATRAN neutral file has a configuration number in its element data. It is assumed that it relates to the element's analysis type. PATRAN elements without a configuration number use the FEM Builder analysis type that is specified in the user interface. Non-zero configuration numbers are assumed to be the analysis characteristic code as used by FEM Builder. This implies that all elements supported in Fem Builder are supported in the interface.

Material Properties

The following table lists the PATRAN supported material properties and their supported counterparts in Fem Builder.

PATRAN	Fem Builder
Elastic Modulus Poisson's Ratio Shear Modulus	Elastic Modulus – Isotropic Elastic Modulus - Orthotropic Elastic Modulus – Anisotropic
Thermal Expansion 6 coefficients	Expansion – Isotropic Expansion – Orthotropic Expansion – Anisotropic
Thermal Conductivity 6 conductivities	Conductivity – Isotropic Conductivity – Orthotropic Conductivity – Anisotropic
Density	Density
Specific Heat	Specific Heat – CP
Emissivity	Emissivity

Coordinate Systems

The interface reads a coordinate system's origin, points on Z and X axis, rotation matrix, and type (e.g. rectangular, cylindrical, spherical).

Loads and Boundary Conditions

For structural results, the following table lists PATRAN supported loads and their supported counterparts in Fem Builder.

Type	PATRAN	Fem Builder
Structural	Translations	Restraints (1-3)
	Forces	Nodal Forces (1-3)
	Pressure	Pressure
	Temperature	Temperature

Constraints

The interface reads constraints (MPC data) from PATRAN into Fem Builder.

Groups

The interface reads group id, name, member's type, and member's label.

Points

The interface reads point label, and coordinates.

Curves

The interface reads curve label, and curve coefficients.

Top

PATRAN Write Neutral File Interface

This interface writes a PATRAN neutral file and files containing results from a finite element model in FEM Builder. The neutral file is the only way to pass data into PATRAN's database. It is organized into "packets" containing the data for a fundamental unit of the model. The following table shows the packets that are supported in this interface.

Packet Number	Packet Description
25	File title
1	Node data
2	Element data
3	Material properties
5	Coordinate system
6	Distributed loads – pressures
7	Node forces
8	Node displacements
14	MPC data – constraints
21	Named components – groups

Nodes

The interface writes node label, coordinates, and label of displacement system.

Elements

The following table lists each supported PATRAN element geometry type and the corresponding Fem Builder element geometry type that supports it. The element configuration number that is output is the FEM Builder element analysis type.

PATRAN Geometry Type	Number of nodes	FEM Builder Geometry Type
Bar	2	Linear Line
	3	Quadratic Line
Triangle	3	Linear Triangle
	6	Quadratic Triangle
	9	Not supported
Quad	4	Linear Quad
	8	Quadratic Quad
	12	Not supported
Tetra	4	Linear Tetra
	10	Quadratic Tetra
Wedge	6	Linear Wedge
	15	Quadratic Wedge
	24	Not supported
Hex	8	Linear Brick
	20	Quadratic Brick
	32	Not supported

Material Properties

The following table lists the PATRAN supported material properties and their supported counterparts in Fem Builder.

PATRAN	Fem Builder
Elastic Modulus Poisson's Ratio Shear Modulus	Elastic Modulus – Isotropic Elastic Modulus - Orthotropic Elastic Modulus – Anisotropic
Thermal Expansion 6 coefficients	Expansion – Isotropic Expansion – Orthotropic Expansion – Anisotropic

Thermal Conductivity 6 conductivities	Conductivity – Isotropic Conductivity – Orthotropic Conductivity – Anisotropic
Mass Density	Density
Specific Heat	Specific Heat – CP
Emissivity	Emissivity

Coordinate Systems

The interface writes a coordinate system's origin, points on Z and X-axis, rotation matrix, and type (e.g. rectangular, cylindrical, spherical).

Loads and Boundary Conditions

For structural results, the following table lists PATRAN supported loads and their supported counterparts in Fem Builder.

Type	ANSYS	Fem Builder
Structural	Translations	Restraints (1-3)
	Forces	Nodal Forces (1-3)
	Pressure	Pressure
	Temperature	Temperature

Constraints

The interface writes constraints (MPC data) from Fem Builder into the neutral file format.

Groups

The interface writes group id, name, member's type, and member's label.

PATRAN Results Files

The interface writes a file containing a set of results uniquely identified by the result type and set id. A unique file name is generated based on the FEM Builder result id and type. For example, a set of nodal results that have the same result id (i.e. file=2, step=1, and increment=1) would be written to a file named *model_f2s1i1.nres*. A result file containing element results would have an extension of *.eres*. If only one file has been read into the model, then file is not included in the generated file name. The following table lists the FEM Builder nodal and element results that are written to file.

Nodal Result	Displacement
	Temperature
Element Nodal Result Centroidal Result	Stress
	Strain
	Von-Mises stress
	Hydrostatic stress
	Max. shear stress

Top

Return to Table of Contents

Sinda Read Output File Interface

This interface reads Sinda output files and extracts temperature, percent resin reacted, and resin viscosity results if they are defined. The model the results are being added to ***must*** have existing elements.

Sinda Write Input File Interface

This interface writes a Sinda input file. This interface also creates a thermal network model in the FEM Builder document with a model name built from original model name with a suffix of "*_Net*". The "_Net" model consists of nodes and line elements representing the thermal capacitors and conductors of the thermal network. The Sinda input file is written from this thermal network model. The following SINDA input blocks are written in the SINDA input file:

- TITLE
- NODE
- SOURCE (Only if heat sources are present)
- CONDUCTOR
- CONSTANT
- ARRAY
- EXECUTION
- VARIABLES 1
- VARIABLES 2
- OUTPUT CALLS

Capacitors Data

The volume of planar elements is computed from the element area and a user-specified thickness. The volume of axisymmetric elements is computed as the element area revolved around the Z-axis.

Conductors Data

There are three conductor options: actual, area correction, and length correction.

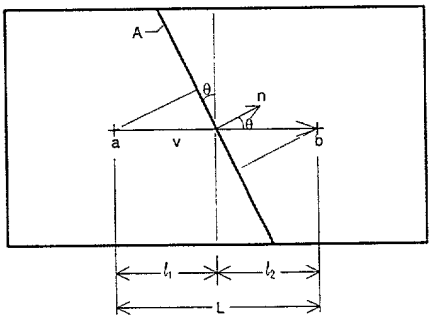


Figure 1: *Conductor Geometry. Points a and b are the element centroids, v is a vector from a to b, n is the common element face normal, and θ is the angle between v and n. L is the distance between a and b.*

The actual option uses the actual area, A, and length, L, in the calculation of the conductance. The area correction option uses the actual length and attempts to correct the area between two elements by using the area indicated by the dotted line in Figure 1. The length correction option uses the actual area and corrects the length by using the sum of the perpendicular distance to the common element face, as shown by the dashed lines in Figure 1. The following table lists how the conductors will be modified for a standard diffusion conductor.

Option	Conductance
Actual	$k\frac{A}{L}$
Area Correction	$k\frac{A\cos\theta}{L}$
Length Correction	$k\frac{A}{L\cos\delta}$

Control Constants

Appropriate control constants will be output depending upon the execution solution procedure selected by the users. If a transient execution procedure is selected, the following control constants are output:

- TIMEND
- OUTPUT
- DTIMEH
- DTIMEI
- DTIMEL
- ARLXCA
- DRLXCA
- ATMPCA
- DTMPCA
- CSGFAC
- SIGMA
- NLOOP

If a steady-state execution procedure is selected, the following control constants are output:

- BALENG
- ARLXCA
- DRLXCA
- SIGMA
- NLOOP

Array Data

All material property data is written as arrays. If a necessary material property is not defined, a skeleton array will be added with a commented line indicating that the properties still need to be defined. Time dependent boundary conditions are also written as array data.

Execution Block

Transient or steady-state execution methods can be selected. Available transient methods are

- FWDBKL
- SNADE
- SNFRDL
- SNDURF

Available steady-state Methods include

- SNDSNR
- SCROUT
- STDSTL
- SNHOSS
- SNSOSS

Return to Table of Contents

TexChem

TexChem is a chemical equilibrium program developed in part under the Air Force Service Life Prediction Technology (SLPT) program. TexChem solves for chemical equilibrium for a specified set of chemical species. FEM Builder supports model definition, boundary condition creation, and specification of chemical species, reactions and associated material properties. Interfaces for reading and writing TexChem files are described in the following sections.

Read TexChem File Interface

This interface reads TexChem spatial result files for analysis results and TexChem command, model, and material files for model definition. Specie concentrations from the results file are saved as analysis results that may be used in postprocessing.

Write TexChem File Interface

This interface writes TexChem command, model, and material files. Prior to writing TexChem files the user should have defined material properties, created a model, and applied boundary conditions. TexChem solves the chemical equilibrium equations for a specified set of chemical species. If temperature is a problem variable, it should be defined as a specie with appropriate boundary conditions. Required material properties, boundary conditions, and user input are described in the following sections.

Material Properties

TexChem uses the following material properties.

1. Chemical species (**required**)
   - Specie name
2. Chemical specie reaction data (**optional**)
   - Reaction name.
   - Reaction formula that identifies reactant and product species and stoichiometric coefficients. For example: 2(H2) + O2 -> 2(H2O). If the stoichiometric coefficient is 1, neither the coefficient nor the parentheses around the specie name are required.
3. Material dependent properties (**required**)
   - Density
   - Specific Heat C_P
   - Isotropic conductivity
   - Chemical Specie Diffusivity: D_{Ref}, D_0, D_1, E_A as used in the following equation.

     $$D = D_{Ref}(D_0 + D_1 C)e^{\left(\frac{E_A}{RT}\right)}$$

   - Chemical Specie Solubility: S_{Ref}, E_A as used in the following equation.

     $$S = S_{Ref}e^{\left(\frac{E_A}{RT}\right)}$$

   - Chemical Reaction Rates are required if reactions are defined: R_{Ref}, R_0,R_1,E_A,Exp_1,Exp_2,...Exp_n as used in the following equation.

     $$R = R_{Ref}(R_0 + R_1 T)e^{\left(\frac{E_A}{RT}\right)}\prod_i C_i^{Exp_i}$$

Boundary Conditions

Boundary conditions should be defined for each chemical specie (or temperature) in the solution as desired. Three types of boundary conditions may be defined: Fixed (Dirichlet), Flux (flux into the body is positive), and Convective (mixed).

User Input

The **Write, TexChem** file interface prompts the user for the following information.

- Command file name
- Solution method
- Solution parameters
  - Absolute tolerance ATol
  - Relative tolerance RTol
  - Maximum number of steps
  - Spatial output times
  - History output node list
- Initial/Constant temperature
- Initial conditions

The write TexChem input interface generates element face groups for each unique boundary condition value and type and then writes the finite element grid, boundary conditions, material properties, and user input as described above to the specified command, model, and material files. The names of the model and material files are generated by removing the ".cmd" extension from the command file and appending ".model", and ".material" respectively.

Return to Table of Contents

Material Properties

Material properties are an integral part of FEM. The material properties that must be defined depends on the analysis program being used. The currently supported material properties are listed in Appendix – Material Properties. Fem Builder supports the ability to create, edit, list, and delete material properties, as well as the ability to read and write material property files, and specify element material orientations. If a material property file includes some properties that are not in the FEM database, those files are treated as a special category called user defined properties.

A material can be created with or without associating a material id, and defining its properties. If the user does not define material id(s) and/or properties, they may do so when editing the material. It is important to note that a material may be associated with multiple material ids. However, a material id can be associated to one and only one material.

Create

The menu bar **Create, Material Property** option is used to create new materials and their properties. The name of the material to create and the element material Ids to associate with the material are specified. For convenience, properties are grouped into classes. Property classes include structural, thermal, chemical, and user defined. User defined properties may be edited but not created.

Upon selecting a property type, the user can enter the property values. The user must select the ***Apply*** button in order to save the new values of a property before changing property types. When a property has been created for the specified material a checked box will appear next to it in the list of supported properties.

If the user is in create mode and selects an existing property, any changes are treated as if in edit mode.

Temperature dependent properties are supported. In order to add additional temperature dependent values to the property, the user places the mouse over the grid containing the properties and presses the right most mouse button. This brings up a popup menu that provides insert, append, and delete functionality for additional temperature dependent property input.

Edit

The menu bar **Edit, Material Property** option is used to edit existing materials and their properties. An existing material may be selected from the pop-down menu in the Edit Material dialog box, and the list of material ids associated with the specified material may be modified.

The user may create properties or may edit any existing property, including user defined property types. Existing properties contain a checked box next to it in the property list. After a property is added or redefined the user must press the ***Apply*** button in order to save the changes before selecting a different property type.

Temperature dependent property values can be added in the same way that they are in the **Create, Material Property** dialog.

List

This interface is used to list the current material properties in the database to the list window. The user is prompted to specify which materials and their associated properties are to be displayed. The specified material(s) and associated property definitions are displayed in the list window.

Delete

This interface is used to delete materials and their existing properties. The user is able to delete any material and associated properties that is created or read into FEM Builder, including user-defined

The **Reset** button will reset the number of levels and range back to the optimized estimate given by the program.
The **Advanced** button opens the **Modified Area** dialog. This option computes the first and last contour value such that the area between the range minimum and the first contour and the area between the range maximum and the last contour are approximately those specified by the user. This improves the visibility of contours where stresses approach infinity (i.e. cracks).
The **Location** option allows the user to locate the legend display on either side of the view.

Deformation

The Deformation tab contains a check box that allows the user to specify whether they want to view deformation. If so, the user can select a scaling factor. The user may specify an actual scale factor or a percent factor based on the view.

The **actual scale factor** scales the deformation by directly multiplying the factor and deformation.
The **percent factor** scales the deformation according to the given percent of the view.

Display

The Display tab allows the user to modify the display of the deformed geometry. These modifications include line style, display type, and color.

The **multiple color** option will use the default color of the model.
The **single color** option will allow the user to select a color from the color dialog.
When in the single color option, the color dialog is activated by pressing the color button.

Top

Vector Plots

Result

The Result tab allows the user to select a set of results for display. The user will specify a result type, a set of results and the component of the result to be displayed. When contouring a CAXA model the user can specify the CAXA angle. The title edit box gives the user the opportunity to customize the title that will appear above the contour plot.

Deform

The Deform tab contains a check box that allows the user to specify whether they want to view deformation. If so, the user can select a scaling factor. The user may specify an actual scale factor or a percent factor based on the view.

The **actual multiplication factor** scales the deformation by directly multiplying the factor and deformation. The **percent of view factor** scales the deformation according to the given percent of the view.

Vector

The Vector tab allows the user to modify the vector lengths on the vector plot. The proportional check box allows the user to specify whether vector scaling is proportional to corresponding result values or is based on the current view.

The **scaling** option is the percentage of the vector length relative to the corresponding results or the total view depending on whether or not proportion is checked.

Display

The Display tab allows the user to modify the display of the deformed geometry. These modifications include line style, display type, and color.

The **multiple color** option will use the default color of the model.
The **single color** option will allow the user to select a color from the color dialog.
When in the single color option, the color dialog is activated by pressing the color button.

Top

XY Plots

XY plots are an important way of looking at result data. FEM Builder currently supports Time History plots and Analysis Result plots. Modify Current Plot options give access to parameters controlling how result data is displayed.

Time History Plots

This interface is used to generate time history plots. They are generated using all the analysis result sets of a specified result type and component at the selected nodes or element centroids in the current model. The option to average allows the user to average element results at nodes. The y-axis is defined by the models result data. The x-axis is defined by time. If no times are present, the result set ids are used for the x-axis values. The user may plot the results of the specified entities or the maximum, minimum, sum, or average of the set of selected entities.

Analysis Results Plots

This interface is used to generate analysis result plots. They are generated using the specified result type, set, and component at the selected nodes or element centroids in the current model. The option to average allows the user to average element results at nodes. The y-axis is defined by the selected result set and component. The x-axis is defined using one of the following three methods: ordered set of points, coordinate axis, and curve(s).

The ordered set of points method projects the selected entities onto the invisible line segment(s) and uses the relative distance between the projected entities to define the x-axis. The coordinate axis method uses the coordinates of the selected entities relative to the user specified coordinate system (default is the global system) or the work plane coordinates. The user specified axis of the selected system is used to define the x-axis. The curve(s) method projects the selected entities onto the user specified curve(s) and uses the curve parameters of the projected points to define the x-axis.

Top

Return to Table of Contents

Tools

Coincident Node Check

This function allows a finite element model to be checked for coincident nodes. The coincident node tolerance is the radial distance from the node. If another node falls within this distance, the nodes are considered to be coincident. When merging coincident nodes, occurrences of the node with the higher label are renamed to the lower label and then the higher label is deleted.

In the Check Coincident Nodes dialog, the user enters the coincident node tolerance and clicks the Check button. The number of Coincident Nodes will be displayed in the dialog box. To view details about the coincident nodes, the user can click on the List button. The coincident nodes will be displayed in the List Information window in the following format.

Coincident node check

Keep,	Delete
2	6
3	7
5	9

Any coincident nodes are merged when the Merge button is pressed.

Element Distortion Check

This function checks the relative distortion of elements using a user selection from among several methods. The methods are Gauss Point Distortion, Aspect Ratio, Area Ratio, Quad Angle, Triangle Angle, and Element Warping. Each element is checked by breaking it up into its component faces and checking each face for errors.

Gauss Point Distortion: This function checks FEM element gauss integration points. The checks are similar to checks made by ABAQUS but not identical since the integration points are not the same.

An error message is issued saying that the element gauss point area or volume is very small or negative if the determinant of the Jacobian at one or more gauss integration points is very small or negative.

For quadrilateral and brick elements, an angle check between isoparametric lines is made at each integration point. A warning message is issued if an angle less than 45° or greater than 135° is found.

Aspect ratio: This is not the traditional aspect ratio. Simply stated, it is the longest side of a face divided by the shortest side.

Area ratio: The area of a face is calculated and then divided by the ideal area. (A square is the ideal for quadrilateral. A 45° right triangle is the ideal shape for a triangle.)

Quad Angle: Angles are generated for each corner of all quadrilaterals and compared to the ideal. (90 degrees).

Triangle Angle: Angles are generated for each corner of all triangles and compared with the ideal. (90, 45, 45 degrees)

Note: The order of the angles does matter; for instance a (45, 45, 90) will produce an error.

Element Warping: A normal vector is calculated at each corner of the element. For quadrilaterals, two angles are calculated for the angles between each set of opposing corners' normal vectors. For triangles, three angles are calculated for the angle between the normal vectors of each set of adjacent corners.

After choosing Element Distortion Check in the Tools menu, a dialog will come up with a list of functions to check the distortion of the model with a corresponding default error tolerance for each function. The Gauss Point Distortion will be flagged as a default.

The Default Error Tolerances Are:

Aspect Ratio	>	5.0
Area Ratio	<	0.6
Quad Angle	outside 90° (+/-)	20.0°
Triangle Angle	outside 45° (+/-)	10.0°
Element Warping	>	15.0°

After choosing the methods and their corresponding error tolerances, test the model by pressing OK to perform the tests.

An error message will be written to the List Box for any elements that fail at least one of the selected distortion tests. Messages include any selected tests in the following format:

Label Gauss Method 1 Gauss Method 2 Any other methods that were chosen

Label	Gauss pt Angle	Gauss pt Area/Volume	AspectR	AreaR	QAngle	TAngle	Warp
5	Angle < 45 or > 135	Ok	1.202	0.543	153.435	45.000	0.000
8	Ok	Ok	3.162	0.682	90.000	71.565	0.000

The user can use the listed values to compare with the error tolerances to determine under which criterion the test failed.

Options

The following options allow the user to set preferences on mouse picking, work plane use, element defaults, subdivision, and unit systems. Changing these options will ***only*** effect future related operations in Fem Builder.

Mouse

Picking entities, or locating positions on the view with the mouse are two ways of defining coordinate data. A pick hierarchy is followed to search for possible entity matches. The hierarchy looks for—**points, nodes, curve end points, curve intersections**, and finally **screen location.** Points which fall within the pick zone are considered first. Nodes, curve end points, and curve intersections are then considered. If no matches are found, a point on the work plane will be located. The user can select any combination of these entity types in order to control what the pick zone will identify and snap to when picking or locating on the view.

The **pick aperture** (pick zone) can be made larger or smaller. Its size is determined by setting a percent of the current view. The visible aperture will not change although the pick aperture's size changes.

The **select adjacent curves automatically** option is useful when the user has to pick many adjoining curves. It can be toggled on or off for all future curve picking operations.

Work Plane

The work plane is used to simplify geometry modeling. For example, when locating points with the mouse, screen pick points are projected onto the work plane. If the work plane is perpendicular to the view plane, a mouse click will not return coordinates since the projection point is undefined.

By default, the work plane is initially located on the *xy* plane of the global rectangular coordinate system. However the work plane can be either **rectangular** or **cylindrical** and moved to any position. A rectangular work plane allows input of *x, y, z* coordinates; a cylindrical work plane allows the input of r, θ, z coordinates. The **grid snap** option controls the coordinates received from mouse clicks. The user can specify the grid size increment and whether screen picks snap to work plane grid points.

The **use work plane coordinates** option allows the user to specify if coordinates entered are in global or work plane coordinates.

Element Defaults

The element defaults will be used to fill the dialog box for element creation or editing.

The **Element Order** option selected from a pop-down menu sets up whether the ordering of nodes in the element is linear or quadratic.

The **Analysis Type** sets up a default analysis to correspond with the applicable geometry type. Geometry types are divided into 4 'families': linear, planar, shell and solid. Each family has a pop-down list of analysis types. Once selected, any geometry type of that family will have the selected default analysis type.

The **Material ID** and **Property ID** boxes allow the user to input an integer value for these defaults.

Subdivision

The user can set up the defaults for several methods of selecting default curve subdivision.

The **Global element size** option uses a percentage of the model size. The program default is 10%.
The **Deviation from curve** option has not been implemented.
The **Maximum aspect ratio** option places an upper limit on aspect ratio. The program default is 5.
The **Minimum elements per curve** option will subdivide the curve into at least this many divisions. The program default is 2.
The **Minimum quad/brick angle** option has not been implemented.
The **Mapped variation** option uses a percentage of the length of a side of a mapped mesh to determine how the subdivisions should take place. The program default is 10%.

Units

The Units dialog gives the user a pop-down menu of choices for **Unit System**. If the user selects one of the pre-defined unit systems, the corresponding units for length, force, mass, time and temperature are loaded into the **Units** table in un-editable form. In the case of a **User Defined** unit system, all of the values in the Units table are available for editing.

Return to Table of Contents

View

Display Settings

The display settings include entity **Visibility**, **Geometry** display settings, **Fem** display settings, and **General** settings. The **Visibility** page allows the user to set the visibility of geometry, fem entities, and boundary conditions for the specified model. The visibility of the entire model can be changed as well. The **Geometry** and **Fem** pages allows the user to toggle their entity labels on or off, change their default color, and modify the symbol or line style that is used to represent the entity in the model. The **General** page allows the user to turn on or off the visibility of the header which includes the Fem Builder version and the date and time.

On the **Fem** page, elements have some unique display settings. They are element display, shrink, and orientation. The element display types are described as follows. The **Line Draw** display type draws all element edges. The **Free Face** display type draws all edges of 2D elements and faces of 3D elements not shared by two elements. The **Free Edge** display type draws edges of 2D and 3D elements not shared by two elements. The **Material Edge** display type draws edges of 2D and 3D elements not shared by two elements with the same material ID. The **Hidden Line** display type draws a display with hidden lines removed. All of the color fill display types generate displays with hidden lines removed, visible faces are filled with color, and the specified element edges are displayed. The user can turn on or off the visibility of the element orientation via its checkbox.

Dynamic View

The view origin, size, and orientation can be modified dynamically using the mouse. The cursor mode indicates the function to be performed in response to cursor movement and mouse button use.

The **Pan** cursor mode allows the user to move the model with respect to the screen with the mouse. The pan operation begins when the left mouse button is depressed and terminates when the left mouse is released.

The **Magnify** cursor mode allows the user to redefine the view scale with the mouse. Scaling begins when the left mouse button is depressed. Scaling occurs as the cursor is dragged up or down on the screen. The model increases in scale as the mouse is dragged /pulled down the screen, and decreases as the mouse is dragged/pushed up the screen. Scaling terminates when the left mouse is released.

The **Rotate** cursor mode allows the user to rotate the model with respect to the view center with the mouse. The rotation operation begins when the left mouse button is depressed and terminates when the left mouse is released.

The **3D Rotate** cursor mode allows the user to perform a 3D rotation of the model with respect to the view center with the mouse. The rotation operation begins when the left mouse button is depressed and terminates when the left mouse is released. The rotation operation is most easily described as follows.

The **Zoom** cursor mode allows the user to define the view bounds by drawing the view rectangle with the mouse. The first corner of the view rectangle is defined when the left mouse button is depressed. The rectangle is drawn as the cursor is dragged, i.e. the cursor is moved with the left mouse button depressed, across the screen. The view rectangle is complete and the view is redrawn when the left mouse button is released. If the left mouse button is clicked, i.e. depressed and released without moving the cursor, the mouse point will become the view center with no change to the view scale.
Consider the view to contain a ball centered in the view with the same diameter as the minimum view dimension. Pressing the left mouse button when the cursor is over the ball is like touching the ball. Dragging the cursor rotates the ball with respect to the view center. Releasing the left mouse button releases the touch on the ball. If the left mouse button is pressed outside the radius of the ball, a 2D rotation is performed.

The **Reset** function scales the model to fit in the view with the default orientation.

The **Autoscale** function scales the model to fit in the view with the current orientation.

Orientation

The view orientation can be altered via keyboard input using the **Eye Direction**, **Screen Rotate**, and **Model Rotate** options. The **Eye direction** option requests the X,Y,Z vector that corresponds to the direction normal to the screen. The **Screen Rotate** option requests the rotation angles about the screen Z,Y',X" axes. In a similar fashion, the **Model Rotate** option requests the rotation angles about the model Z,Y',X" axes.

Dialog position

The position of interactive dialogs can be specified as **Left**, **Right**, or **Center** with respect to the main program window. This setting is persistent between interactive sessions as is the location and size of the main window and list box.

Status flags

The **Dynamic** flag indicates whether the view should be redrawn repeatedly during pan, magnify, and rotate operations or just once at the end of the operation.

The **List Box** flag indicates whether the list box should be visible or not. FEM Builder outputs information to the list box to indicate the status of operations.

The **Toolbar** and **Status Bar** flags control the visibility of the same.

Return to Table of Contents

Overview

FEM Builder is an interactive program written by Thiokol Propulsion to provide Finite Element Modeling (FEM) tools for building and analyzing finite element models, and to provide interfaces between FEM systems and analysis programs.

Fem Builder has a user interface similar to most PC programs with a menu bar, a toolbar, and a status bar. The program allows users to have one or more documents open at the same time. A document can contain one or more finite element models. Each document can have multiple views.

File

File functions allow the user to open, close, and save FEM Builder files. Read and write functions are provided to read files from and write files to various analysis programs. Others are provided to read and write material property files. The file translation option translates data from one file type to another without saving that information in a FEM Builder document, which can save time if no other FEM Builder functions are to be performed.

Create

Creation functions allow the user create FEM entities, geometry, groups, material properties, meshing, boundary conditions, Chemical Species and Reactions.

Edit

Edit functions allow the user to edit FEM entities, geometry, material properties, and element orientations.

Delete

Delete functions delete selected FEM entities, geometry, material properties, Chemical Species and Reactions from the current model.

List

List functions write selected FEM entities, geometry, element orientations, constraint equations, groups, boundary conditions, analysis results, or material properties, information to the list window.

Post-Processing

Post-processing functions include result superposition, and result interpolation, deformed geometry, contour plots, vector plots, and XY plots.

Model Information

Most FEM Builder functions operate on one finite element model, the current model. The Model Information dialog allows the user to set the current model, the model name, and the model title. The dialog also lists other model-related information.

Tools

Tools include coincident node check, element distortion check, and customizable default options. The default options are entity picking, work plane use, element defaults, subdivision, and units.

View

The display settings are used to set the visibility and defaults of geometry entities, fem entities, and the view header. The view orientation can be specified directly via keyboard input. The view origin, size and orientation can also be modified dynamically using dynamic view. Status flags indicate the visibility of the toolbar, status bar, and list box.

Window

These window functions are the same functions found in many other PC programs. The **New Window** function creates a new view of the model that can be scaled, rotated, and relocated independently of any other view. The **Cascade**, **Tile**, and **Arrange Icon** options all rearrange existing views.

Help

The help option provides access to program version information.

Program Defaults

The coordinate system used by FEM Builder is the same used by most analysis programs. Interpretation of coordinates depends on the element being referenced. For example, for plane elements coordinates 1,2 are X, Y, for axisymmetric elements coordinates 1,2 are R, Z, and for solid elements coordinates 1,2,3 are X, Y, Z.

Default model units are Inches, pound mass, pound force, second, and degrees Fahrenheit.

Return to Table of Contents

Keyword: ChemicalReactions

The *ChemicalReactions card defines chemical reactions. If chemical reactions are defined, this section should follow the chemical specie input and precede the material property input. The heat of reaction is assumed to be zero if not defined.

Example:

```
*ChemicalReactions
**Name,Heat or reaction,Units / Formula
 Salt
 Na + Cl -> NaCl
 Water
 2(H2) + O2 -> 2(H2O)
```

Keyword: Material

The *Material card defines a material. All succeeding properties will be associated with this material until the next *Material card. The material name is a required parameter.

Example:

```
*Material, Aluminum
 ...
*Material, Steel
 ...
```

Keword: Property

The *Property card is used to define a property of a material. The property name is a required parameter. A *Material card must precede the *Property card. Properties are tables of numeric or character values. All property fields are comma delimited. The first card after the *Property card is a record containing property value names. The next card may contain property values or property value units. If any field in the record is numeric, the record is assumed to contain property values. All succeeding records until the next keyword card contain property values.

Example:

```
*Property, Elastic Isotropic
      E,      Nu,     Temp
    PSI,    IN/IN,       F
 10300000., 0.3300000, 70.000000
 9900000.0, 0.3300000, 200.00000

*Property, Density
      Rho,     Temp
LBM/IN-IN-IN,       F
 0.101000000, 70.000000
```

Keyword: Attribute

The *Attribute card is used to associate parameters and values with properties or materials. The *Attribute card is followed by one or more records containing parameters names, values and units.

Example:

```
*Attribute
 TRef = 70.000000, F
```

Keyword: Note

The *Note card is used to associate text with properties or materials. The note ends at the next keyword card.

properties. If all the properties are deleted for this material name the user will be asked if the material should be deleted as well.

Read

This interface is used to read material properties from a material property (.pdb extension) file or from a Fem Builder (.fdb extension) file. It reads any property that is formatted correctly (see Appendix – Material Properties), including property notes and attributes. This is how user-defined properties are brought into FEM Builder. Upon reading the file, the user is prompted to select the material properties they would like to add to their current Fem Builder model.

Write

This interface is used to write material properties to a file. The property file's default name is the current model name with a .pdb extension. It writes all properties that are in the current model's database.

Return to Table of Contents

Appendix – Material Properties

Material Property Classes

Class Name
Structural
Thermal
User-Defined

Currently Supported Material Properties

Property Name	Property Value Names
Elastic Isotropic	E, Nu, Temp
Elastic Orthotropic	E1, E2, E3, Nu12, Nu13, Nu23, G12, G13, G23, Temp
Elastic Anisotropic	E1111, E1122, E2222, E1133, E2233, E3333, E1112, E2212, E3312, E1212, E1113, E2213, E3313, E1213, E1313, E1123, E2223, E3323, E1223, E1323, E2323, Temp
Expansion Isotropic	A, Temp
Expansion Orthotropic	A1, A2, A3, Temp
Expansion Anisotropic	A11, A12, A22, A13, A23, A33, Temp
Conductivity Isotropic	K, Temp
Conductivity Orthotropic	K1, K2, K3, Temp
Conductivity Anisotropic	K11, K12, K22, K13, K23, K33, Temp
Density	Rho, Temp
Specific Heat Cp	Cp, Temp
Specific Heat Cv	Cv, Temp
Specific Heat Ratio	Gamma, Temp
Emissivity	Epsilon, Temp
Chemical Reaction Rates	Reaction, Rate, R0, R1, EA, Exp1, Exp2, Exp3, Exp4, Exp5, Exp6
Chemical Specie Diffusivity	Specie, DRef, D0, D1, EA
Chemical Specie Solubility	Specie, SRef, EA

Material Property File Format

The material property file is a keyword-formatted file. Keyword cards begin with an asterisk in column one. Keywords are not case sensitive. Keyword cards may have an associated parameter. The parameter is separated from the keyword by a comma. A comment is a line with asterisks in columns one and two. Comments and blank lines may be placed anywhere in the file. Comments and blank lines are ignored on input. Other than the keyword starting in column one, the file is a free format file. If a line ends with a comma, the next line is assumed to be a continuation line. Recognized keywords include *Material, *Property, *Attribute, and *Note and are discussed below.

Keyword: ChemicalSpecies

The *ChemicalSpecies card defines chemical species. If chemical species are defined, this section should precede the material property input. The molecular weight is assumed to be zero if not defined.

Example:
*ChemicalSpecies
**Name,Molecular weight
Specie1

Keyword: ChemicalReactions

The *ChemicalReactions card defines chemical reactions. If chemical reactions are defined, this section should follow the chemical specie input and precede the material property input. The heat of reaction is assumed to be zero if not defined.

Example:

```
*ChemicalReactions
**Name,Heat or reaction,Units / Formula
 Salt
 Na + Cl -> NaCl
 Water
 2(H2) + O2 -> 2(H2O)
```

Keyword: Material

The *Material card defines a material. All succeeding properties will be associated with this material until the next *Material card. The material name is a required parameter.

Example:

```
*Material, Aluminum
 ...
*Material, Steel
 ...
```

Keword: Property

The *Property card is used to define a property of a material. The property name is a required parameter. A *Material card must precede the *Property card. Properties are tables of numeric or character values. All property fields are comma delimited. The first card after the *Property card is a record containing property value names. The next card may contain property values or property value units. If any field in the record is numeric, the record is assumed to contain property values. All succeeding records until the next keyword card contain property values.

Example:

```
*Property, Elastic Isotropic
      E,      Nu,     Temp
    PSI,    IN/IN,       F
 10300000., 0.3300000, 70.000000
 9900000.0, 0.3300000, 200.00000

*Property, Density
      Rho,     Temp
LBM/IN-IN-IN,       F
 0.101000000, 70.000000
```

Keword: Attribute

The *Attribute card is used to associate parameters and values with properties or materials. The *Attribute card is followed by one or more records containing parameters names, values and units.

Example:

```
*Attribute
 TRef =  70.000000, F
```

Keword: Note

The *Note card is used to associate text with properties or materials. The note ends at the next keyword card.

Example:
*Note
Material properties extracted from TWR-15995
Space Shuttle RSRM Nozzle Materials Data Book

Unit Specification Rules

Units specified for property sets and attributes are simply text strings stored with the properties. However, unit conversion is possible when the unit descriptions follow the conventions described below.

Property units are expected to consist of the following:

Unit Type	Unit Names
Length	IN, FT, M, CM, MM
Force	LBF, LB, N, DYNE
Mass	LBM, KG, GRAM, SLUG
Time	S, SEC, MIN, HR
Temperature	F, R, K, C
Pressure	PSI, PSF, PA, KSI, MSI, MPA, ATM
Energy	BTU, CAL, J, KJ, KCAL, ERG
Power	W, WATT, KW
Other	1, MOL, MOLE

Rules for unit specification are as follows. The unit text is expected to have at most one /. Any parenthesis encountered on input will be ignored. All * and / operators, other than the first /, will be replaced with a -. No other operators are recognized. Examples of some common properties are shown below.

Property	Units
Elastic Modulus	PSI
Poisson's Ratio	IN/IN
Expansion Coefficient	IN/IN-R

Return to Table of Contents

Geometry Creation

FEM Builder supports the creation of coordinate systems, points, curves, surfaces and volumes. A consistent input method for coordinates and vectors helps in creating geometry quickly. A work plane is also utilized to aid in the definition of many of these geometric entities.

Coordinate and Vector Input
Work Plane
Coordinate System
Groups
Point
Curve
Surface
Volume

Coordinate and Vector Input

Coordinate definition is fundamental to geometry modeling. Coordinates are needed directly to define points and curves, and indirectly to define surfaces and volumes. A uniform method is implemented in Fem Builder to simplify coordinate definition.

Entering values, picking, or locating are used to define coordinates. Entering two or three values from the keyboard is simple and straightforward. While this is not a particularly exciting or efficient option, it works. Pick coordinate definition gets the coordinates of the entity picked. Picking a point, or node, gets the coordinates. Curve end points and curve intersections can also be picked. Locate retrieves the coordinates of a mouse click on the work plane.

A filter is provided to control the action of the mouse. The pick filter allows the user to specify which entities to look for—points, nodes, curve end points, and curve intersections. Mouse location and grid snap can also be toggled on or off.

In pick or locate mode, a hierarchy is followed to search for possible matches. When the program is in *'Get Coordinate'* mode, a small box is drawn around the cursor pick point. Points which fall within the pick zone are considered first. Nodes, curve end points, and curve intersections are then considered. If no matches are found, a point on the work plane will be located. If the grid snap is on, only points on the grid are allowed.

Coordinates can be entered or viewed in either global coordinates or in the work plane coordinates. Global coordinates are always (x,y,z). The work plane coordinate labels depend on the work plane type—rectangular or cylindrical.

Vectors or directions are also needed for several geometry functions. Defining an extrusion or an axis of revolution often will require a vector. A uniform method is implemented in Fem Builder to simplify vector definition. Vectors can be defined by entering the vector manually, by specifying two points (using the coordinate functions described above), by selecting a coordinate axis, or by a curve tangent. Vectors can be entered or viewed in either the global coordinates or in the work plane coordinates, just as points.

Top

Work Plane

The work plane is used to simplify geometry modeling. For example, when locating points with the mouse, screen pick points are projected onto the work plane. If the work plane is perpendicular to the view plane, a mouse click will not return coordinates since the projection point is undefined. The work plane is always defined but does not have to be used. For example, when keying in coordinates, a checkbox is displayed to allow the coordinates to be entered in global coordinates or work plane coordinates.

By default, the work plane is initially located on the xy plane of the global rectangular coordinate system. The work plane can be placed on a coordinate system plane, or moved to any position. The work plane is either rectangular or cylindrical. A rectangular work plane allows input of x,y,z coordinates; a cylindrical

work plane allows the input of r,θ,z coordinates. The work plane also has a grid snap option that controls the coordinates received from mouse clicks. The user can specify the grid size increment and whether screen picks snap to work plane grid points.

Top

Coordinate System

Coordinate systems can be rectangular, cylindrical, or spherical. Specifying the system origin, a point on the positive x-axis, and a point on the xy or xz plane creates a rectangular coordinate system. Specifying the system origin, a point on the positive z-axis, and a point on the rz-plane creates a cylindrical or spherical coordinate system.

Top

Point

A point defines a location in three-dimensional space. Points can be created by picking points, nodes, curve end points, curve intersections, by screen location on the work plane using the mouse, or by entering coordinates. Other point create functions create points between two coordinate locations, or on a curve.

Top

Curve

A curve is a line, arc, circle, or spline defined in a plane or in general three-dimensional space.

Specifying end points creates lines. Lines can be created from pairs of coordinates using the Single option or between successive points using the Point-to-Point option.

There are four different methods to create an arc. These methods are three points on the arc, the center-start-end points, the start-end points and the radius, and the center-start points and the angle. The start-end-radius arc and the center-start-angle arc definitions use the work plane to provide the additional data to complete these arc definitions. The start-end-radius arc and the center-start-angle arc use the work plane normal vector (z-axis) and the right-hand rule to define the plane and circumscribed direction of the arc.

A circle function is available which creates a circular arc from the center point and a radius. It also uses the work plane normal vector to complete the circle definition.

Splines are a series of connected cubic segments with second derivative continuity (C^2). Splines are defined by specifying points along the spline and two end conditions. There are two types of possible end conditions: zero curvature or specified tangents. The zero curvature end conditions are the default. Specified tangent end conditions may be input using the vector input options described above.

Curves may also be created with the offset function, which creates a curve based on the selected curve, an offset direction, and an offset distance.

Top

Surface

A surface is a region bounded by a curve or curves in two- or three-dimensional space. Surface creation functions include boundary curves, extrusion, and revolution. The Boundary function creates a surface by specifying all the boundary curves. The Extrude function creates a surface by extruding a set of connected curves through a given distance. The Revolve function creates a surface by revolving a set of connected curves about an axis of revolution through a specified angle.

Future surface functions will include other creation options such as sweep, loft and mesh of curves, as well as split and join.

Top

Volume

A volume is a three-dimensional space bounded by a surface or surfaces. Volume creation functions include boundary surfaces, extrude, and revolve. The Boundary function creates a volume by specifying all surfaces that enclose the volume. The Extrude function creates volume using a surface and an extrusion direction and distance. The Revolve function creates a volume using a surface with an origin, an axis of revolution, and an extrusion angle.

Future volume function will include sweep, split and Boolean operations.

Top

Groups

FEM Builder supports the creation of point, curve, surface, and volume groups. Unlike node and element groups, these groups are not supported by and may not be imported through the Ideas Master Series interface. The group creation functions allow selection of the points, curves, surfaces, or volumes after which a group name may be specified. If the specified name already exists, it will be replaced.

Top

Return to Table of Contents

Create Finite Element Model Entities

FEM Builder supports the creation of nodes, elements, constraint equations, groups, and boundary conditions.

Node
Element
Constraint equations
Mesh generation
Groups
Boundary Conditions

Node

The same coordinate input method used for points is also used for node coordinate input.

Top

Element

Default values for element creation are loaded from the defaults in the registry, which can be set up with the Tools/Options selections from the menu. The program automatically assigns an element label. Then the user can modify the geometry type, the analysis type, the Material ID, the Property ID, the hybrid flag, the reduced flag and the nodes that comprise the element. The Apply button and the OK button will create the new element with the values from the dialog box.

Node connectivity allows the user to select the nodes that will comprise the element. A diagram of the node order is displayed to assist in the assignment of the nodes.

Top

Constraint Equations

Constraint generation provides a simple way for users to create constraint equations in a finite element model. Constraint equations are a way to control specific behavior and relationships between nodes in a finite element model. The constraint equations created by this function bind elements together that do not share common nodes. Only the three displacement degrees of freedom (DX, DY, and DZ) are used by the constraint generation functions.

The constraint generation wizard is activated from the Fem Builder menu by selecting Create and then Constraint. In the *Constraint Wizard* dialog box, the user specifies the constraint generation method, selects the desired degrees of freedom, and enters necessary tolerances. The *zero coefficient tolerance* is used to eliminate near zero coefficients from the constraint equations. The *move node tolerance* is used to indicate the maximum allowed distance to move a node. The next page in the Constraint Wizard is node selection where the nodes to constrain are indicated. If constraining elements are required, these elements are then selected. The last page of the Constraint Wizard is a summary of the options selected. Pressing the Finish button generates the constraint equations.

Three methods of generation constraint equations are available in Fem Builder:

- Constrain nodes to elements
- Constrain nodes to free faces
- Constrain nodes to adjacent nodes

Constrain Nodes to Elements

When nodes are constrained to elements, the element that contains the node to constrain is found. The location of the node relative to the element natural coordinate system is determined. The element shape

functions are evaluated at this natural coordinate position. The zero coefficient tolerance is used to eliminate near zero shape function values, and the shape function values are normalized. Using the displacement coordinate system for each node, the shape function value associated with that node undergoes a coordinate system transformation. The results of this transformation are the coefficients of the constraint equation. These coefficients are again compared with the zero coefficient tolerance to eliminate near zero coefficients, and the constraint equation is created.

Constrain Nodes to Element Free Faces
When nodes are constrained to element free faces, the closest element free face to the node that is being constrained is found. The location of the node relative to the element natural coordinate system is determined, and the node is moved to be on the nearest free face of the element if this new location is within the move node tolerance. If not, no constraint equation is generated. The element shape functions are evaluated at this natural coordinate position. The zero coefficient tolerance is used to eliminate near zero shape function values, and the shape function values are normalized. Using the displacement coordinate system for each node, the shape function value associated with that node undergoes a coordinate system transformation. The results of this transformation are the coefficients of the constraint equation. These coefficients are again compared with the zero coefficient tolerance to eliminate near zero coefficients, and the constraint equation is created.

Constrain Nodes to Adjacent Nodes
When nodes are constrained to adjacent nodes, the adjacent nodes that form the shortest line that, within tolerance, includes the node to be constrained are found. The node to be constrained must be within the move node tolerance to be moved. If so, the node is then moved to the closest position on the line. If no such line exists, then the node is not moved and no constraint equation is generated. The parametric coordinate of the node to constrain relative to the shortest line is determined. A linear interpolation function is evaluated at this parametric coordinate to obtain shape function values, similar to those of a linear line element. The zero coefficient tolerance is used to eliminate near zero shape function values, and the shape function values are normalized. Using the displacement coordinate system for each node, the shape function value associated with that node undergoes a coordinate system transformation. The results of this transformation are the coefficients of the constraint equation. These coefficients are again compared with the zero coefficient tolerance to eliminate near zero coefficients, and the constraint equation is created.

Top

Groups

FEM Builder supports the creation of node, element, and element face groups. Node and element groups are supported by and may be imported through the Ideas Master Series interface. However, element face groups may not be imported through the Ideas Master Series interface. Element face groups are useful in creating boundary conditions since the precise set of faces to be loaded may be selected. The group creation functions allow selection of the nodes, elements, or faces after which a group name may be specified. If the specified name already exists, it will be replaced.

Top

Boundary Conditions

FEM Builder supports the creation of many types of boundary conditions. Boundary conditions have been grouped into three categories: structural, thermal, and chemical. Each is applied to one or more FEM entities and is identified by a load set id or specie name. Some of the boundary conditions have reference temperatures associated to them. There are four methods of defining boundary conditions: constant, tabular, interpolated, and isentropic flow. The following table summarizes the currently supported boundary conditions, their parameters, and their available definition methods.

Type	Applied to	Identification	Ref. Value	Available Methods
Pressure	Element free faces	Set ID	No	Constant, Interpolate, Isentropic
Restraint	Nodes	Set ID	No	Constant, Interpolate

Force	Nodes	Set ID	No	Constant
Moment	Nodes	Set ID	No	Constant
Temperature	Nodes	Set ID	No	Constant
Convection	Element free faces	Set ID	Yes	Constant
Radiation	Element free faces	Set ID	Yes	Constant
Heat Flux	Element free faces	Set ID	No	Constant
Point Source	Nodes	Set ID	No	Constant
Volume Source	Elements	Set ID	No	Constant
Specie Concentration	Element free faces	Specie name	No	Constant
Specie Convection	Element free faces	Specie name	Yes	Constant
Specie Flux	Element free faces	Specie name	No	Constant

The restraint boundary condition has six degrees of freedom. The user can select one of six types to define which degrees of freedom are restrained.

1. Specify – restrain whatever the user selects.
2. X Symm – restrain translation in the x-direction and rotation about the y and z-axes.
3. Y Symm – restrain translation in the y-direction and rotation about the x and z-axes.
4. Z Symm – restrain translation in the z-direction and rotation about the x and y-axes.
5. Pinned – restrain translation in the x, y and z-directions.
6. Fixed – restrain translation in the x, y and z-directions, and rotation about the x, y and z-axes.

In order to create chemical species boundary conditions, the user must first define (create) the chemical specie to be applied as a boundary condition.

Constant

This method of boundary condition creation applies to all types of boundary conditions. Boundary conditions are created in a two-step process:

1. Selection of the nodes, elements, or element faces to which the BC will be applied.
2. Specification of the BC set ID and value or values.

Tabular

Not yet implemented.

Interpolated

The interpolation method of boundary condition creation is used for creating enforced restraints based on the deformation of an existing structural model and for extracting pressure loads from CFD models. Selected nodes or the centroid of selected element faces are projected onto the grid that has the specified analysis result in a manner similar to that used for result interpolation. Boundary conditions are created in a two-step process:

1. Selection of the nodes or element faces to which the BC will be applied.
2. Specification of the model and result to interpolate from, the maximum projection distance, and the BC set ID.

Isentropic flow

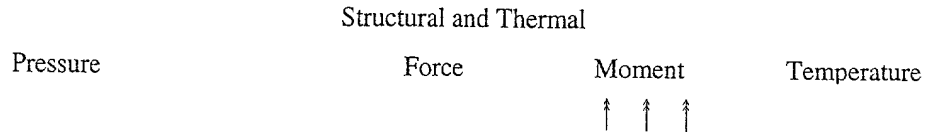
The isentropic flow method of boundary condition creation is used for creating pressure loads based on isentropic flow equations. Boundary conditions are created in a two-step process:

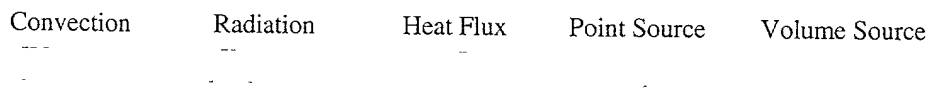
1. Selection of the element faces to which the BC will be applied.
2. Specification of the BC set ID, throat radius, throat Z location, stagnation pressure, the ratio of specific heats (gamma), and the nozzle exit direction (increasing or decreasing Z).

Symbols

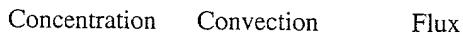
Each of the boundary condition types has a symbol used to represent it in the model. They are as follows:

Structural and Thermal





Species



Top

Return to Table of Contents

Create Finite Element Model Entities

FEM Builder supports the creation of nodes, elements, constraint equations, groups, and boundary conditions.

Node
Element
Constraint equations
Mesh generation
Groups
Boundary Conditions

Node

The same coordinate input method used for points is also used for node coordinate input.

Top

Element

Default values for element creation are loaded from the defaults in the registry, which can be set up with the Tools/Options selections from the menu. The program automatically assigns an element label. Then the user can modify the geometry type, the analysis type, the Material ID, the Property ID, the hybrid flag, the reduced flag and the nodes that comprise the element. The Apply button and the OK button will create the new element with the values from the dialog box.

Node connectivity allows the user to select the nodes that will comprise the element. A diagram of the node order is displayed to assist in the assignment of the nodes.

Top

Constraint Equations

Constraint generation provides a simple way for users to create constraint equations in a finite element model. Constraint equations are a way to control specific behavior and relationships between nodes in a finite element model. The constraint equations created by this function bind elements together that do not share common nodes. Only the three displacement degrees of freedom (DX, DY, and DZ) are used by the constraint generation functions.

The constraint generation wizard is activated from the Fem Builder menu by selecting Create and then Constraint. In the *Constraint Wizard* dialog box, the user specifies the constraint generation method, selects the desired degrees of freedom, and enters necessary tolerances. The *zero coefficient tolerance* is used to eliminate near zero coefficients from the constraint equations. The *move node tolerance* is used to indicate the maximum allowed distance to move a node. The next page in the Constraint Wizard is node selection where the nodes to constrain are indicated. If constraining elements are required, these elements are then selected. The last page of the Constraint Wizard is a summary of the options selected. Pressing the Finish button generates the constraint equations.

Three methods of generation constraint equations are available in Fem Builder:

- Constrain nodes to elements
- Constrain nodes to free faces
- Constrain nodes to adjacent nodes

Constrain Nodes to Elements

When nodes are constrained to elements, the element that contains the node to constrain is found. The location of the node relative to the element natural coordinate system is determined. The element shape

functions are evaluated at this natural coordinate position. The zero coefficient tolerance is used to eliminate near zero shape function values, and the shape function values are normalized. Using the displacement coordinate system for each node, the shape function value associated with that node undergoes a coordinate system transformation. The results of this transformation are the coefficients of the constraint equation. These coefficients are again compared with the zero coefficient tolerance to eliminate near zero coefficients, and the constraint equation is created.

Constrain Nodes to Element Free Faces
When nodes are constrained to element free faces, the closest element free face to the node that is being constrained is found. The location of the node relative to the element natural coordinate system is determined, and the node is moved to be on the nearest free face of the element if this new location is within the move node tolerance. If not, no constraint equation is generated. The element shape functions are evaluated at this natural coordinate position. The zero coefficient tolerance is used to eliminate near zero shape function values, and the shape function values are normalized. Using the displacement coordinate system for each node, the shape function value associated with that node undergoes a coordinate system transformation. The results of this transformation are the coefficients of the constraint equation. These coefficients are again compared with the zero coefficient tolerance to eliminate near zero coefficients, and the constraint equation is created.

Constrain Nodes to Adjacent Nodes
When nodes are constrained to adjacent nodes, the adjacent nodes that form the shortest line that, within tolerance, includes the node to be constrained are found. The node to be constrained must be within the move node tolerance to be moved. If so, the node is then moved to the closest position on the line. If no such line exists, then the node is not moved and no constraint equation is generated. The parametric coordinate of the node to constrain relative to the shortest line is determined. A linear interpolation function is evaluated at this parametric coordinate to obtain shape function values, similar to those of a linear line element. The zero coefficient tolerance is used to eliminate near zero shape function values, and the shape function values are normalized. Using the displacement coordinate system for each node, the shape function value associated with that node undergoes a coordinate system transformation. The results of this transformation are the coefficients of the constraint equation. These coefficients are again compared with the zero coefficient tolerance to eliminate near zero coefficients, and the constraint equation is created.

Top

Groups

FEM Builder supports the creation of node, element, and element face groups. Node and element groups are supported by and may be imported through the Ideas Master Series interface. However, element face groups may not be imported through the Ideas Master Series interface. Element face groups are useful in creating boundary conditions since the precise set of faces to be loaded may be selected. The group creation functions allow selection of the nodes, elements, or faces after which a group name may be specified. If the specified name already exists, it will be replaced.

Top

Boundary Conditions

FEM Builder supports the creation of many types of boundary conditions. Boundary conditions have been grouped into three categories: structural, thermal, and chemical. Each is applied to one or more FEM entities and is identified by a load set id or specie name. Some of the boundary conditions have reference temperatures associated to them. There are four methods of defining boundary conditions: constant, tabular, interpolated, and isentropic flow. The following table summarizes the currently supported boundary conditions, their parameters, and their available definition methods.

Type	Applied to	Identification	Ref. Value	Available Methods
Pressure	Element free faces	Set ID	No	Constant, Interpolate, Isentropic
Restraint	Nodes	Set ID	No	Constant, Interpolate

Force	Nodes	Set ID	No	Constant
Moment	Nodes	Set ID	No	Constant
Temperature	Nodes	Set ID	No	Constant
Convection	Element free faces	Set ID	Yes	Constant
Radiation	Element free faces	Set ID	Yes	Constant
Heat Flux	Element free faces	Set ID	No	Constant
Point Source	Nodes	Set ID	No	Constant
Volume Source	Elements	Set ID	No	Constant
Specie Concentration	Element free faces	Specie name	No	Constant
Specie Convection	Element free faces	Specie name	Yes	Constant
Specie Flux	Element free faces	Specie name	No	Constant

The restraint boundary condition has six degrees of freedom. The user can select one of six types to define which degrees of freedom are restrained.

1. Specify – restrain whatever the user selects.
2. X Symm – restrain translation in the x-direction and rotation about the y and z-axes.
3. Y Symm – restrain translation in the y-direction and rotation about the x and z-axes.
4. Z Symm – restrain translation in the z-direction and rotation about the x and y-axes.
5. Pinned – restrain translation in the x, y and z-directions.
6. Fixed – restrain translation in the x, y and z-directions, and rotation about the x, y and z-axes.

In order to create chemical species boundary conditions, the user must first define (create) the chemical specie to be applied as a boundary condition.

Constant

This method of boundary condition creation applies to all types of boundary conditions. Boundary conditions are created in a two-step process:

1. Selection of the nodes, elements, or element faces to which the BC will be applied.
2. Specification of the BC set ID and value or values.

Tabular

Not yet implemented.

Interpolated

The interpolation method of boundary condition creation is used for creating enforced restraints based on the deformation of an existing structural model and for extracting pressure loads from CFD models. Selected nodes or the centroid of selected element faces are projected onto the grid that has the specified analysis result in a manner similar to that used for result interpolation. Boundary conditions are created in a two-step process:

1. Selection of the nodes or element faces to which the BC will be applied.
2. Specification of the model and result to interpolate from, the maximum projection distance, and the BC set ID.

Isentropic flow

The isentropic flow method of boundary condition creation is used for creating pressure loads based on isentropic flow equations. Boundary conditions are created in a two-step process:

1. Selection of the element faces to which the BC will be applied.
2. Specification of the BC set ID, throat radius, throat Z location, stagnation pressure, the ratio of specific heats (gamma), and the nozzle exit direction (increasing or decreasing Z).

Symbols

Each of the boundary condition types has a symbol used to represent it in the model. They are as follows:

Structural and Thermal

Pressure | Force | Moment | Temperature

Convection | Radiation | Heat Flux | Point Source | Volume Source

Species

Concentration | Convection | Flux

Top

Return to Table of Contents

Mesh Generation

Meshes can be created in FEM Builder on surfaces and volumes. Structured (mapped) and arbitrary (free) meshes are easily generated. Users select a geometric entity to mesh, and specify a mesh type, the mesh parameters, and the curve subdivision. The user may preview the mesh before creating it.

Surface Mesh

A mesh may be generated on a planar or non-planar surface. Linear and quadratic order triangular elements and/or quadrilateral elements are used when meshing surfaces.

Structured Surface Mesh

A mapped mesh is the most straightforward and intuitive method of meshing. The boundary curves of a surface must be collected and divided into four edges. A regular grid is formed on the "four-sided" surface by mapping points on opposite sides together. A structured mesh is a mapped mesh or some hybrid of a mapped mesh. There are four varieties of structured surface meshes that can be generated in FEM Builder: mapped, 1D-transition, 2D-transition type 1, and 2D-transition type 2. Each of these mesh types is shown in Figure 1.

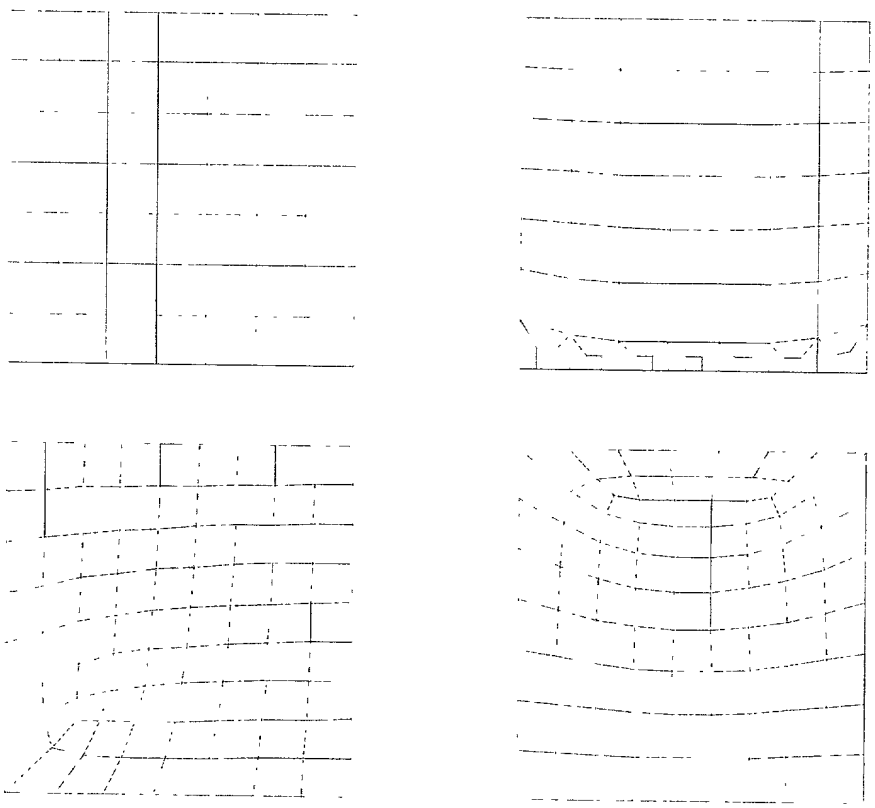


Figure 1: Structured surface meshes: Mapped, 1D-transition, 2D-transition type 1, and 2D-transition type 2

A standard mapped mesh can only be created when opposite sides have equal subdivisions. A 1D-transition mesh modifies a regular mapped mesh by subdividing one row of elements on an edge to obtain finer curve subdivision. Both 2D-transition meshes are constructed from two mapped meshes—one central mesh and a "wrap-around" mesh—to mesh surfaces with irregular subdivisions. This set of structured mesh types will mesh most curve subdivisions.

Arbitrary Surface Mesh

An arbitrary mesh is constructed with an arbitrary number of edges. An intricate process is used to determine the location of interior points and how those points should be connected into elements. Arbitrary meshes generated by FEM Builder are either triangular or quadrilateral. Occasionally, a triangular element is needed in an otherwise quadrilateral mesh. Each of these mesh types is shown in Figure 2.

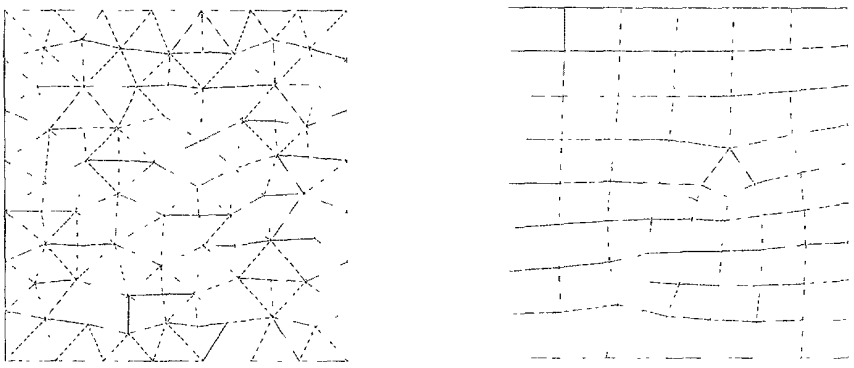


Figure 2: Arbitrary meshes: Triangle and Quadrilateral

Top

Volume Mesh

A mesh may be generated on a volume. Linear and quadratic order brick elements, wedge elements, and/or tetrahedron elements are used when meshing volumes.

Structured Volume Mesh

Similar to a surface-mapped mesh, a volume-mapped mesh is the most straightforward and intuitive method of meshing volumes. The volume to be meshed must be defined by six surfaces, with each surface connected to four surfaces and opposite of the other. A regular grid is formed on the six-sided volume by mapping points on opposite sides together. A structured mesh is a mapped mesh or some hybrid of a mapped mesh. There are four varieties of structured volume meshes that can be generated in FEM Builder: mapped, 1D-transition, 2D-transition type 1, and 2D-transition type 2. These meshes correspond directly with their respective surface mesh counterparts. In the volume mesh version of these transition meshes, the third dimension of the meshes is mapped. This yields two opposite surfaces with transition meshes and the other surfaces with mapped meshes. Each of these mesh types is shown in Figure 3.

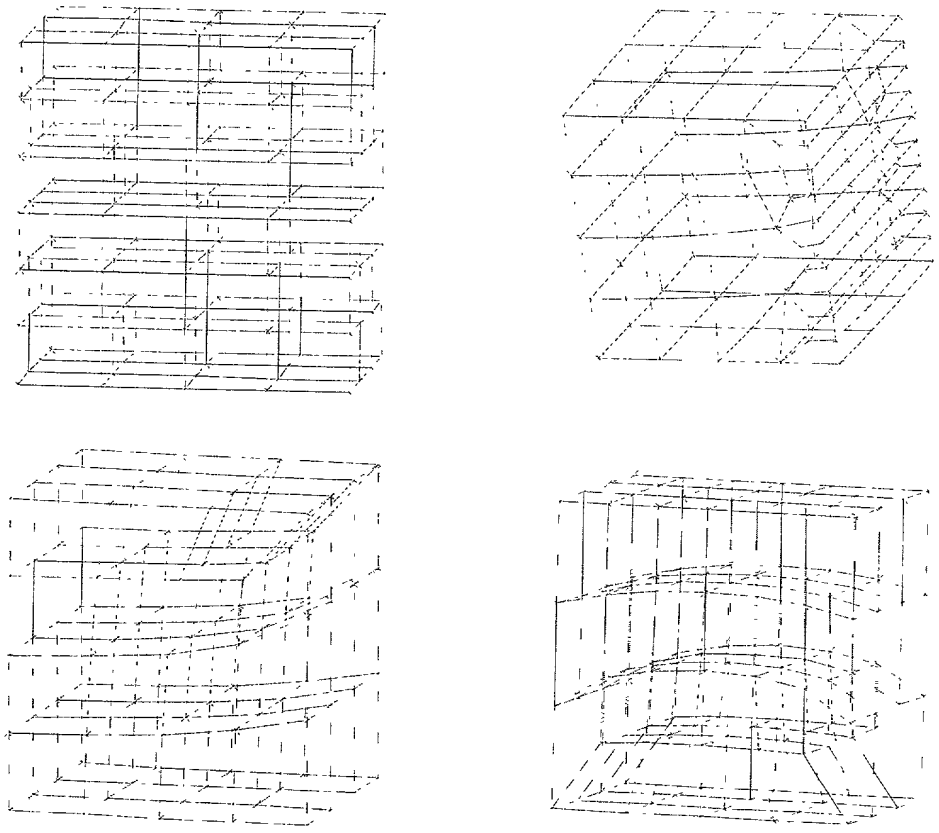


Figure 3: Volume meshes: Mapped, 1D-transition, 2D-transition type 1, and 2D-transition type 2

Arbitrary Volume Mesh
Arbitrary volume meshes have not been implemented yet.

Top

Return to Table of Contents

Mesh Generation

Meshes can be created in FEM Builder on surfaces and volumes. Structured (mapped) and arbitrary (free) meshes are easily generated. Users select a geometric entity to mesh, and specify a mesh type, the mesh parameters, and the curve subdivision. The user may preview the mesh before creating it.

Surface Mesh

A mesh may be generated on a planar or non-planar surface. Linear and quadratic order triangular elements and/or quadrilateral elements are used when meshing surfaces.

Structured Surface Mesh

A mapped mesh is the most straightforward and intuitive method of meshing. The boundary curves of a surface must be collected and divided into four edges. A regular grid is formed on the "four-sided" surface by mapping points on opposite sides together. A structured mesh is a mapped mesh or some hybrid of a mapped mesh. There are four varieties of structured surface meshes that can be generated in FEM Builder: mapped, 1D-transition, 2D-transition type 1, and 2D-transition type 2. Each of these mesh types is shown in Figure 1.

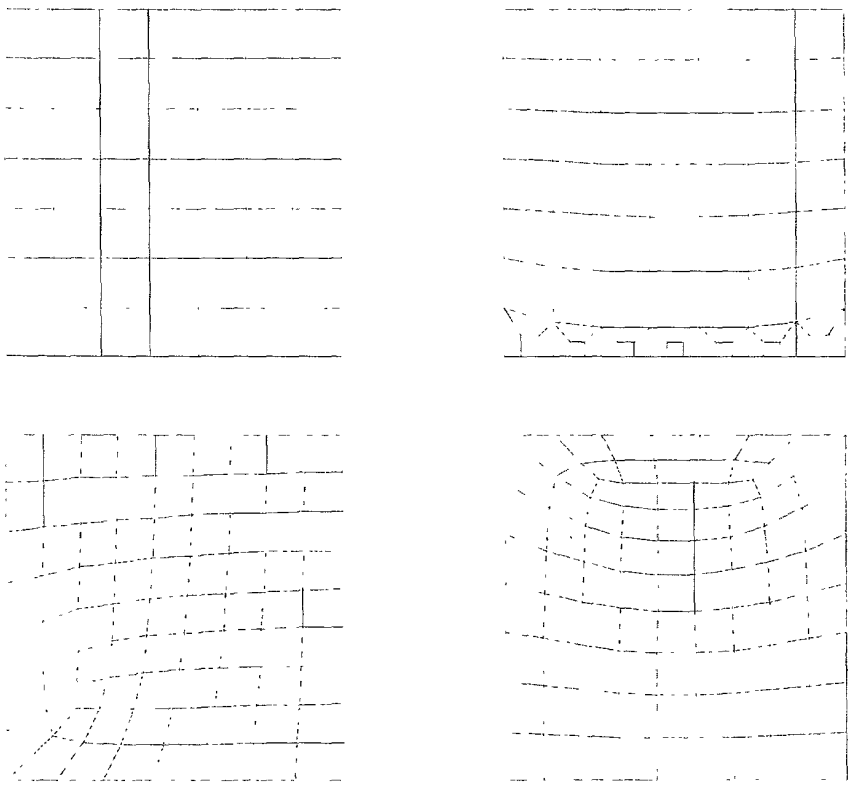


Figure 1: Structured surface meshes: Mapped, 1D-transition, 2D-transition type 1, and 2D-transition type 2

A standard mapped mesh can only be created when opposite sides have equal subdivisions. A 1D-transition mesh modifies a regular mapped mesh by subdividing one row of elements on an edge to obtain finer curve subdivision. Both 2D-transition meshes are constructed from two mapped meshes—one central mesh and a "wrap-around" mesh—to mesh surfaces with irregular subdivisions. This set of structured mesh types will mesh most curve subdivisions.

Arbitrary Surface Mesh

An arbitrary mesh is constructed with an arbitrary number of edges. An intricate process is used to determine the location of interior points and how those points should be connected into elements. Arbitrary meshes generated by FEM Builder are either triangular or quadrilateral. Occasionally, a triangular element is needed in an otherwise quadrilateral mesh. Each of these mesh types is shown in Figure 2.

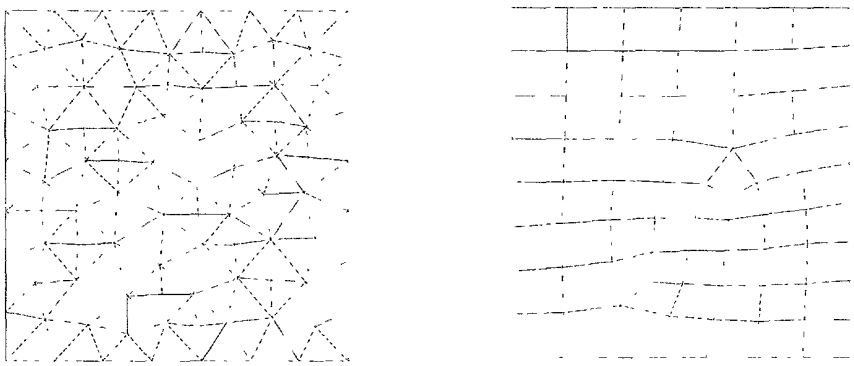


Figure 2: Arbitrary meshes: Triangle and Quadrilateral

Top

Volume Mesh

A mesh may be generated on a volume. Linear and quadratic order brick elements, wedge elements, and/or tetrahedron elements are used when meshing volumes.

Structured Volume Mesh

Similar to a surface-mapped mesh, a volume-mapped mesh is the most straightforward and intuitive method of meshing volumes. The volume to be meshed must be defined by six surfaces, with each surface connected to four surfaces and opposite of the other. A regular grid is formed on the six-sided volume by mapping points on opposite sides together. A structured mesh is a mapped mesh or some hybrid of a mapped mesh. There are four varieties of structured volume meshes that can be generated in FEM Builder: mapped, 1D-transition, 2D-transition type 1, and 2D-transition type 2. These meshes correspond directly with their respective surface mesh counterparts. In the volume mesh version of these transition meshes, the third dimension of the meshes is mapped. This yields two opposite surfaces with transition meshes and the other surfaces with mapped meshes. Each of these mesh types is shown in Figure 3.

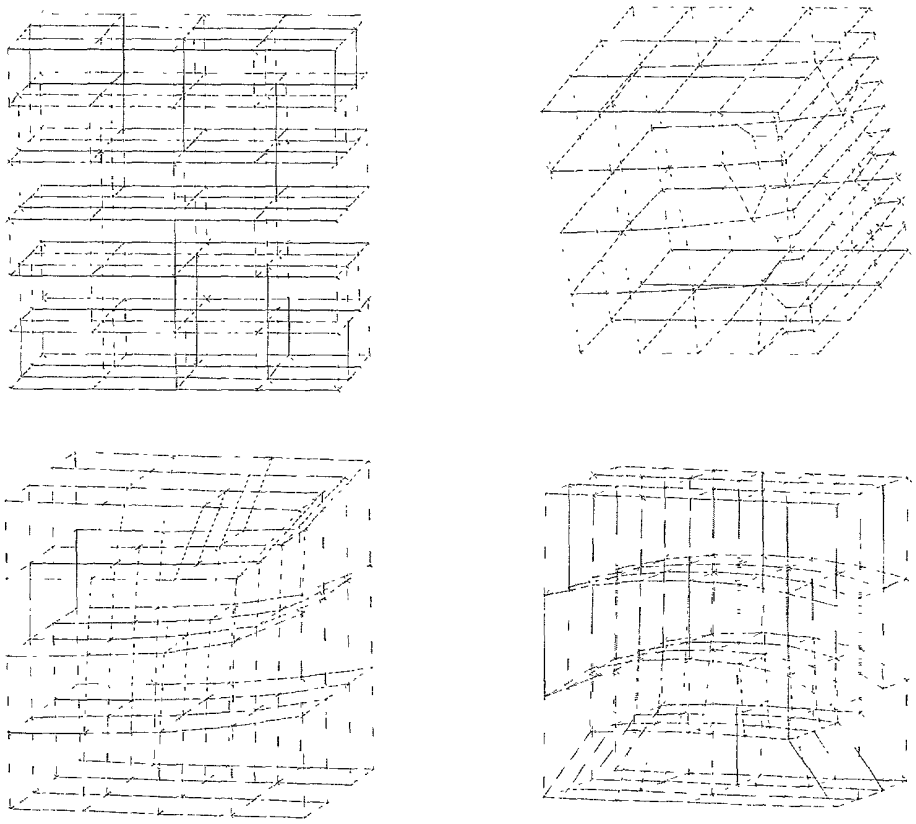


Figure 3: Volume meshes: Mapped, 1D-transition, 2D-transition type 1, and 2D-transition type 2

Arbitrary Volume Mesh
Arbitrary volume meshes have not been implemented yet.

Top

Return to Table of Contents

Material Properties

Material properties are an integral part of FEM. The material properties that must be defined depends on the analysis program being used. The currently supported material properties are listed in Appendix – Material Properties. Fem Builder supports the ability to create, edit, list, and delete material properties, as well as the ability to read and write material property files, and specify element material orientations. If a material property file includes some properties that are not in the FEM database, those files are treated as a special category called user defined properties.

A material can be created with or without associating a material id, and defining its properties. If the user does not define material id(s) and/or properties, they may do so when editing the material. It is important to note that a material may be associated with multiple material ids. However, a material id can be associated to one and only one material.

Create

The menu bar **Create**, **Material Property** option is used to create new materials and their properties. The name of the material to create and the element material Ids to associate with the material are specified. For convenience, properties are grouped into classes. Property classes include structural, thermal, chemical, and user defined. User defined properties may be edited but not created.

Upon selecting a property type, the user can enter the property values. The user must select the ***Apply*** button in order to save the new values of a property before changing property types. When a property has been created for the specified material a checked box will appear next to it in the list of supported properties.

If the user is in create mode and selects an existing property, any changes are treated as if in edit mode.

Temperature dependent properties are supported. In order to add additional temperature dependent values to the property, the user places the mouse over the grid containing the properties and presses the right most mouse button. This brings up a popup menu that provides insert, append, and delete functionality for additional temperature dependent property input.

Edit

The menu bar **Edit**, **Material Property** option is used to edit existing materials and their properties. An existing material may be selected from the pop-down menu in the Edit Material dialog box, and the list of material ids associated with the specified material may be modified.

The user may create properties or may edit any existing property, including user defined property types. Existing properties contain a checked box next to it in the property list. After a property is added or redefined the user must press the ***Apply*** button in order to save the changes before selecting a different property type.

Temperature dependent property values can be added in the same way that they are in the **Create, Material Property** dialog.

List

This interface is used to list the current material properties in the database to the list window. The user is prompted to specify which materials and their associated properties are to be displayed. The specified material(s) and associated property definitions are displayed in the list window.

Delete

This interface is used to delete materials and their existing properties. The user is able to delete any material and associated properties that is created or read into FEM Builder, including user-defined

properties. If all the properties are deleted for this material name the user will be asked if the material should be deleted as well.

Read

This interface is used to read material properties from a material property (.pdb extension) file or from a Fem Builder (.fdb extension) file. It reads any property that is formatted correctly (see Appendix – Material Properties), including property notes and attributes. This is how user-defined properties are brought into FEM Builder. Upon reading the file, the user is prompted to select the material properties they would like to add to their current Fem Builder model.

Write

This interface is used to write material properties to a file. The property file's default name is the current model name with a .pdb extension. It writes all properties that are in the current model's database.

Return to Table of Contents

Chemical Properties

Chemical properties consist of chemical species, chemical reactions, and chemical specie/reaction related material properties (specie diffusivity, specie solubility, and reaction rates). These properties are all interrelated. Chemical reactions define how species react and so may only refer to defined chemical species. The solubility and diffusivity of chemical species depend on the material they are migrating through. Chemical reaction rates may also depend on the surrounding material, e.g. the surrounding material may be a catalyst to the reaction. As a result chemical specie solubility, specie diffusivity, and reaction rates are defined as material properties.

Chemical Species

Chemical species are degrees of freedom in a specie diffusion problem such as those solved by TexChem. For each defined chemical specie, appropriate specie boundary conditions should be created. Chemical specie solubility and diffusivity properties should be defined for each specie in each material.

Chemical Reactions

Chemical reaction's formula defines how species react. As reactions are defined, if the reaction formula refers to undefined chemical species, the user is given the option of creating the undefined specie(s). The reaction formula is in the following form.

$$coef_{R1}(specie_{R1}) + \ldots\ coef_{Rn}\ (specie_{Rn}) \rightarrow coef_{P1}(specie_{P1}) + \ldots\ coef_{Pm}(specie_{Pm})$$

Where $coef_{R1},\ldots coef_{Rn}$: Reactant stoichiometric coefficients
$coef_{P1},\ldots coef_{Pm}$: Product stoichiometric coefficients
$specie_{R1},\ldots specie_{Rn}$: Reactant species
$specie_{P1},\ldots specie_{Pm}$: Product species

If a stoichiometric coefficient is 1, then neither the coefficient nor the parenthesis around the specie name are required.

Return to Table of Contents

Edit Geometry

FEM Builder supports the editing the work plane, points, curves, surfaces and volumes. The same methods in getting coordinate and vector input in the create geometry dialogs are used in the edit geometry dialogs.

Work Plane Surface
Point Volume
Curve

Work Plane

The work plane is used to simplify geometry modeling. For example, when locating points with the mouse, screen pick points are projected onto the work plane. If the work plane is perpendicular to the view plane, a mouse click will not return coordinates since the projection point is undefined. The work plane is always defined but does not have to be used. For example, when keying in coordinates, a checkbox is displayed to allow the coordinates to be entered in global coordinates or work plane coordinates.

By default, the work plane is initially located on the *xy* plane of the global rectangular coordinate system. However, the user has the option of making it **rectangular** or **cylindrical**. The work plane also has a **grid snap** option that controls the coordinates received from clicks of the mouse. The user can specify the grid size increment and whether screen picks snap to work plane grid points. The work plane can be placed on a local coordinate system plane, or moved to any position. The user can edit its **position** by entering vectors for its X and Y axis or by selecting one of three definition methods. They are **select a local coordinate system plane**, **translate** the work plane, and **rotate** the work plane. The work plane is either rectangular or cylindrical. A rectangular work plane allows input of *x*, *y*, *z* coordinates; a cylindrical work plane allows the input of *r*, θ, *z* coordinates.

Top

Point

The edit point **color** option allows the user to select any number of points using one of the selection methods to identify them. On the color page the current default point color has the focus and will be used if the user does not specify another color.

Top

Curve

There are four ways to edit curves. They are Split, Trim/Extend, Subdivision, and Color.

The edit curve **split** option allows the user to select a curve and specify a split location along the curve. The user may specify this point by location (coordinate input, picking, or locating), by specifying a distance along the curve, or by specifying a parametric percentage of the curve. If the location point is not on the curve, it is projected onto the curve and the split is performed at the projected point.

The edit curve **trim/extend** option allows the user to select a curve and specify a trim/extend location. Coordinate input, picking, or locating a point defines the location. If the projected location falls on the curve, the curve endpoint closest to that location is trimmed. If the projected location does not fall on the curve, the curve endpoint closest to that location is extended. Similar to **split,** when picking or locating a trim point it is projected onto the curve. The switch button allows the user to change which end of the curve to trim.

The edit curve **subdivision** option allows the user to select a curve and specify the type of subdivision in relation to meshing. The subdivision types are uniform, end bias, and center bias. The user can also specify the number of elements and the size ratio.

The edit curve **color** option allows the user to select any number of curves using one of the selection methods to identify them. On the color page the current default point color has the focus and will be used if the user does not specify another color.

Top

Surface

The edit surface **color** option allows the user to select any number of surfaces using one of the selection methods to identify them. On the color page the current default surface color has the focus and will be used if the user does not specify another color.

Mesh Attributes requires the user to pick or choose a surface and edit the attributes that that will be used when the surface is meshed.

Top

Volume

The edit volume **color** option allows the user to select any number of volumes using one of the selection methods to identify them. On the color page the current default point color has the focus and will be used if the user does not specify another color.

Mesh Attributes requires the user to pick or choose a volume and edit the attributes that that will be used when the volume is meshed.

Top

Return to Table of Contents

Edit Finite Element Model Entities

FEM Builder supports editing the ...

Node

The edit node **Definition** option allows the user to select any node for editing. The selected nodes will be assigned the displacement system and coordinates as input by the user. On the edit node page the current work plane displacement system has the focus and will be used if the user does not specify another displacement system. The default coordinates are the current coordinates of the node. The Apply button and the OK button will replace the values for the node with the values from the dialog box.

The edit node **Color** option allows the user to select any number of nodes using one of the selection methods to identify them. The selected nodes will be assigned the color selected by the user. On the color page the current default node color has the focus and will be used if the user does not specify another color.

The edit node **Displacement System** option allows the user to select any number of nodes using one of the selection methods to identify them. The selected nodes will be assigned the displacement system selected by the user. On the displacement system page the current work plane displacement system has the focus and will be used if the user does not specify another displacement system.

Top

Element

The edit element **Definition** option allows the user to select any element for editing. The default values are the current values of the element. Upon selection of an element, the user can modify the geometry type, the analysis type, the material ID, the Property ID, the hybrid flag, the reduced flag and the nodes that comprise the element. The Apply button and the OK button will replace the values for the element with the values from the dialog box.

Hybrid elements are primarily intended for use with incompressible and almost incompressible material behavior.

Reduced integration elements use a lower-order integration to form the element stiffness.

The edit element **Color** option allows the user to select any number of elements using one of the selection methods to identify them. The selected elements will be assigned the color selected by the user. On the color page the current default element color has the focus and will be used if the user does not specify another color.

The edit element **Orientation** option allows the user to select any number of elements using one of the selection methods to identify them. The selected elements will be assigned the orientation selected by the user. On the orientation page the current default element orientation has the focus and will be used if the user does not specify another orientation.

The edit element **MID** option allows the user to select any number of elements using one of the selection methods to identify them. The selected elements will be assigned the material ID selected by the user. On the MID page the current default element MID has the focus and will be used if the user does not specify another MID.

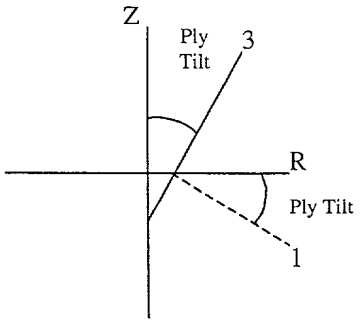
The edit element **PID** option allows the user to select any number of elements using one of the selection methods to identify them. The selected elements will be assigned the property ID selected by the user. On the PID page the current default element PID has the focus and will be used if the user does not specify another PID.

Element Orientation
General material orientation is specified by three Euler angles defining local orientation axes with respect to the global model axes. The first Euler angle is the rotation about Z, the second about Y', and the third about X". This interface is used to define the orientation of materials by defining the element's orientation. The user is prompted to select elements to orient and to define the orientation of those elements. The orientations are associated to the materials by way of the material id assigned in the data of each element.

The orient material function supports orientation of cylindrical and rectangular materials. There are two general methods of defining the orientation: 1) Define all axes at once or 2) define each axis one at a time. The following table shows the ways of defining orientations for each method.

Define all Axes	Define each Axis
Ply tilt angle	Vector directions
Vector directions	Curve tangent (rotating)
Coordinate system	Curve tangent (stationary)
Euler angles	
Element edges	

Ply Tilt Angle
Ply tilt angle orientation implies that a material is cylindrical. The ply tilt angle option prompts the user for the desired ply-tilt angle, which is the angle between the Z-axis and the third material axis. A positive ply tilt angle is a right hand rotation about the Y or Theta axis (see the following figure).



Vector Directions
For the "Define all Axes" method, the user can pick points or enter values to define three vectors for the orientation definition. For the "Define each Axis" method, the user specifies which material axis to define and then pick points or enters a value for the vector used to define the material axis. This interface appears twice for defining each of the first two material vectors. The third is computed using the cross product of the other two.

Coordinate System
The coordinate system option prompts the user to select a coordinate system defined in the current model. It aligns the material axes with the selected coordinate system.

Euler Angles
The Euler angle option prompts the user to specify material orientation via Euler angles. The Euler angles requested are the rotation about the Z-axis, the Y'-axis, and the X"-axis. The primes are used to indicate an axis in its new position after the preceding rotation has occurred.

Element Edge
The element edge option prompts the user to specify which material axis is aligned to which element edge.

Curve Tangent
The curve tangent option prompts the user to select a curve and to specify which material axis, first, second or third, is tangent to the curve. The specified curve may be stationary or rotating. If a rotating curve is specified, the user is prompted for the rotation origin and axis of rotation.

In the cases where vectors are defined, the specified vectors are checked to see if they are orthogonal. If they are not then the cross product of the two specified vectors is used to define the remaining vector. The second vector is redefined using the cross product of the first and third vectors.

Top

Return to Table of Contents

Result Functions

Superposition

Superposition creates new sets of analysis results that are linear combinations of existing analysis sets. The function will either create a result for all of the result types that exist in the selected result sets, or just for selected result types. The multiplication factors to be applied to existing analysis sets must be specified. Default values of analysis set file, step, increment, and mode for the computed results are defined but may be specified.

In the superposition calculation if a result type exists in some analysis sets and not in others, or if results are defined for a node/element in one set and not in others, the result summation will ignore the undefined value.

Top

Interpolation

Interpolation functions allow interpolation of analysis results between grids, i.e. from one model to another, and interpolation to other locations within the same model.

Interpolation between grids

The function to create results by interpolation between grids projects node or centroidal locations from one model onto the grid containing the analysis results. The projection function attempts to find an element that contains the projected point. If an element is located, interpolation to the projected location is performed using the element shape functions. If no element contains the point, the point will be projected onto free faces of elements connected to nodes within a specified distance.

For results on elements at nodes, the function will first project the centroid of the element and then project the element nodes with the restriction that the element the node projects onto must have the same material ID as the centroid projection.

The function has the following steps.

1. The model ***from*** which results will be derived, the model ***to*** which the results will be applied, and the location of the interpolated result are specified.
2. The function requests selection of results to be interpolated on the ***from***-model.
3. Based on the previously specified result location, the function allows selection of the nodes or elements on the ***to***-model to which the result is to be interpolated.
4. The maximum projection distance and resulting analysis set file, step, increment, and set name are specified.

Interpolation to nodes

The function to interpolate results to nodes allows selection of the centroidal or element nodal results to which the function is to be applied. The result is computed by averaging the result from connected elements.

Interpolation to element centroids

The function to interpolate results to element centroids allows selection of the nodal or element nodal results to which the function is to be applied. Interpolation is performed using the element shape functions

Average results on elements at nodes

The function to average results allows selection of the centroidal or element nodal results to which the function is to be applied. The function computes the average result for connected elements with the same material ID. Since the elements to which the node is connected may have different material IDs, the result is stored as an element nodal result.

Top

Return to Table of Contents

Result Display

Deformed Geometry

Result

The Result tab allows the user to select a specific set of displacement results by its set id. When deforming a CAXA model the user can specify the CAXA angle. The title edit box gives the user the opportunity to customize the title that will appear on the screen above the deformed geometry.

Deformation

The Deform tab allows the user to specify a scaling factor for the deformed geometry. The user may specify an actual scale factor or a percent factor based on the view.

The **actual multiplication factor** scales the deformation by directly multiplying the factor and deformation. The **percent of view factor** scales the deformation according to the given percent of the view.

Display

The Display tab allows the user to modify the display of the deformed geometry. These modifications include line style, display type, and color.

The **multiple color** option will use the default color of the model.
The **single color** option will allow the user to select a color from the color dialog.
When in the single color option, the color dialog is activated by pressing the color button.

Undeformed

The Undeformed tab allows the user to modify the display of the undeformed geometry. These modifications include line style, display type, and color.

The **multiple color** option will use the default color of the model.
The **single color** option will allow the user to select a color from the color dialog.
When in the single color option, the color dialog is activated by pressing the color button.

Top

Contour Plots

Result

The Result tab allows the user to select a set of results for display. The user will specify a result type, a set of results and the component of the result to be displayed. When contouring a CAXA model the user can specify the CAXA angle. The user can also specify if they would like to average the results for the same materials in order to create a smoother contour between elements. The title edit box gives the user the opportunity to customize the title that will appear on the screen above the contour plot.

Legend

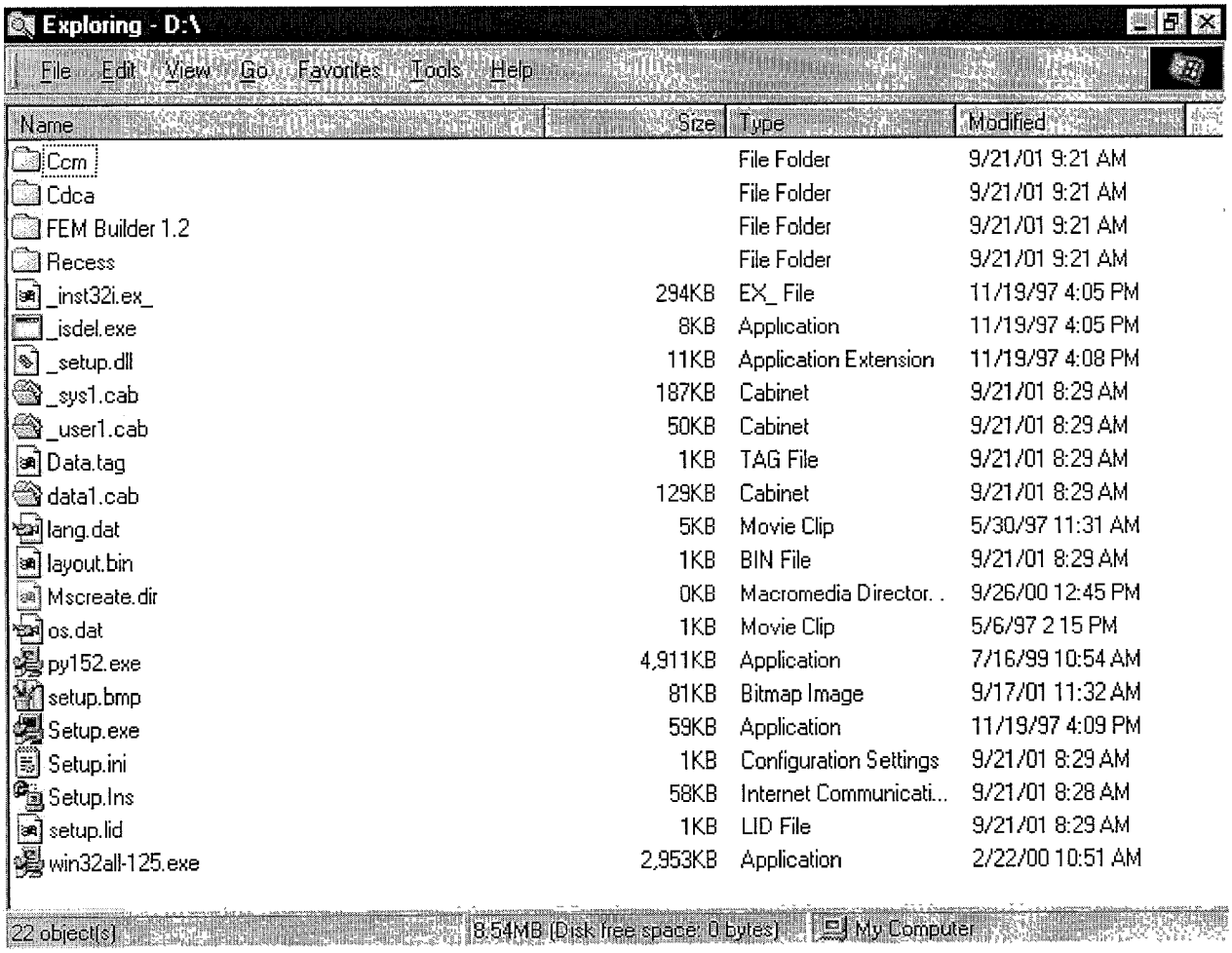
The Legend tab allows the user to specify the number of contour levels to display and their range. The user is given the minimum and maximum result values and an optimized estimate of the number of levels and range. The user can modify the number of levels or the range and increment. The user can enter an actual range or increment or a percent of the range or increment.

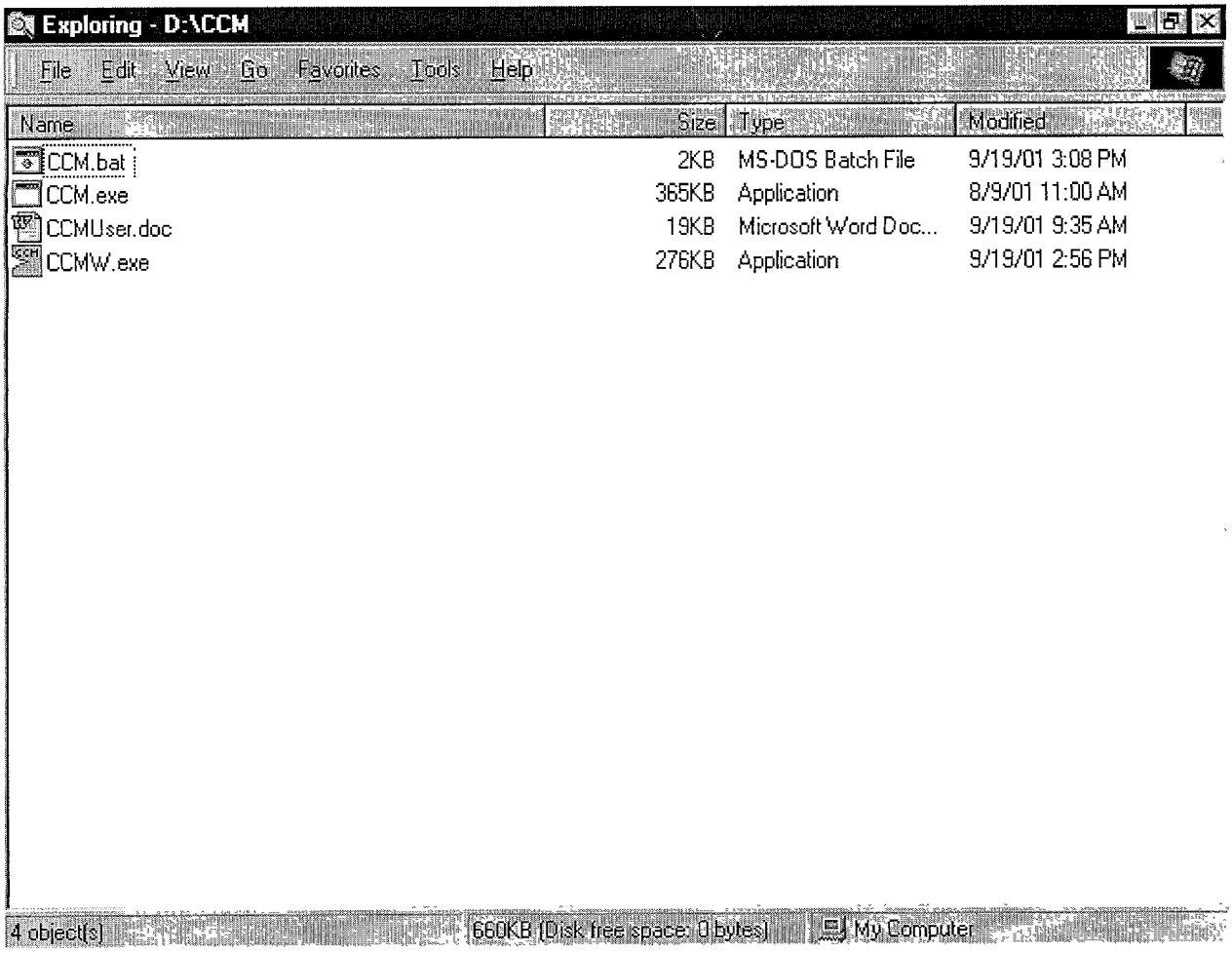
APPENDIX II

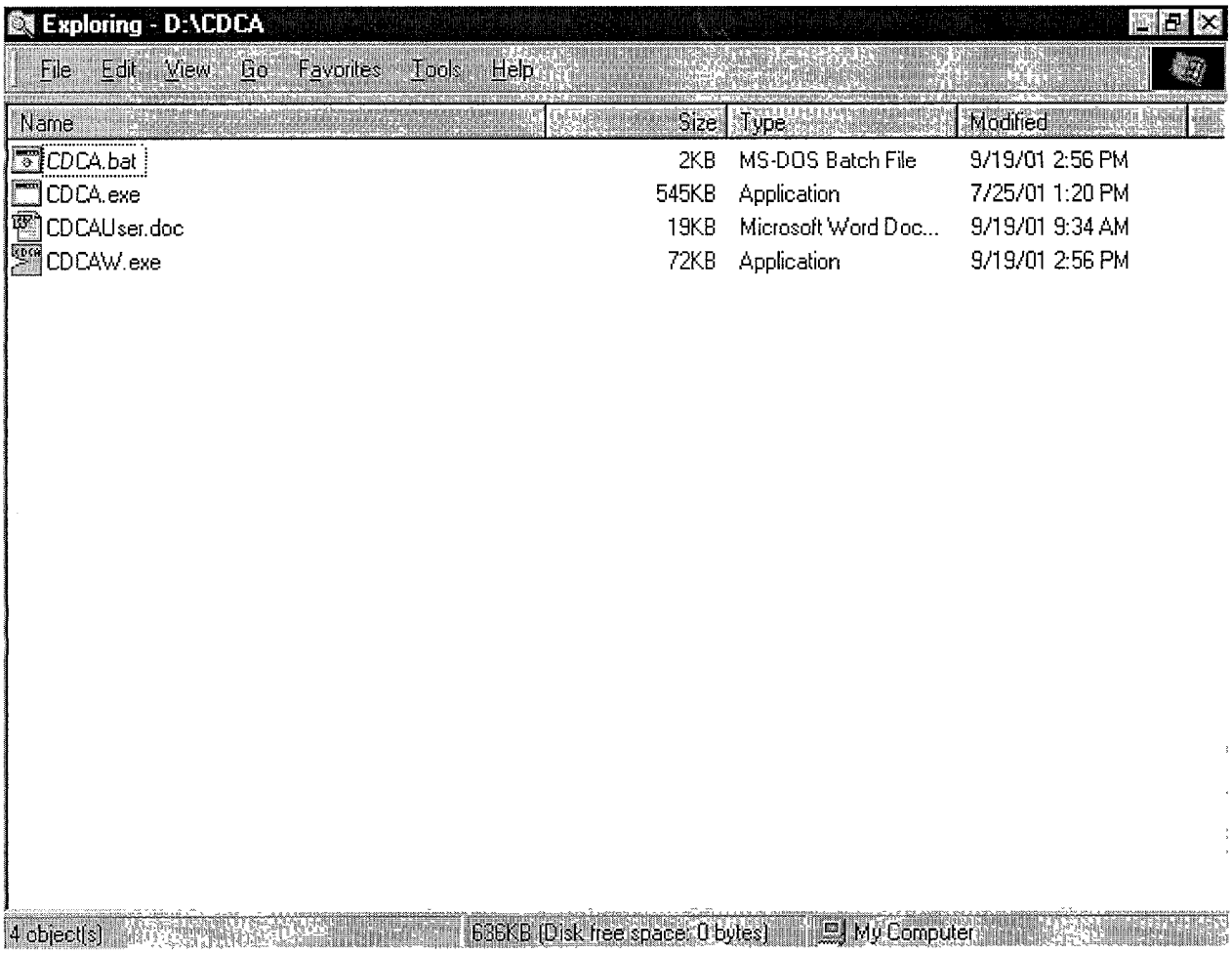
The compact disk attached to the specification and incorporated into the specification by express reference as if fully set forth herein comprises computer software and a computer program listing appendix according to a presently preferred system embodiment and method of the invention. This computer software comprises FEM Builder, which has been built by engineers at Thiokol Propulsion of Brigham City, Utah. The FEM Builder software operates according to the principles of the invention to accomplish coupling of multiple finite analysis programs through the use of a graphical user interface and/or a scripting language.

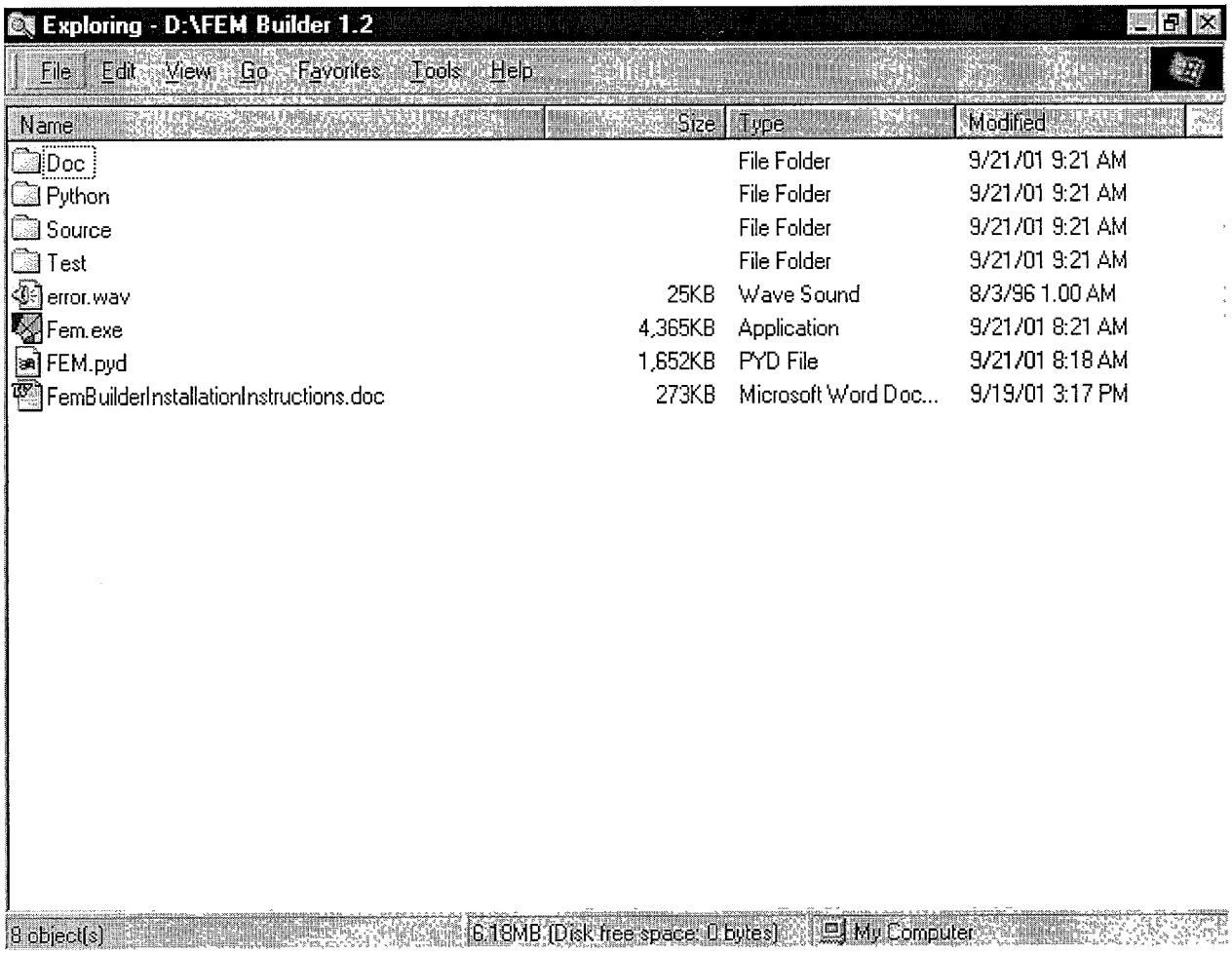
APPENDIX III

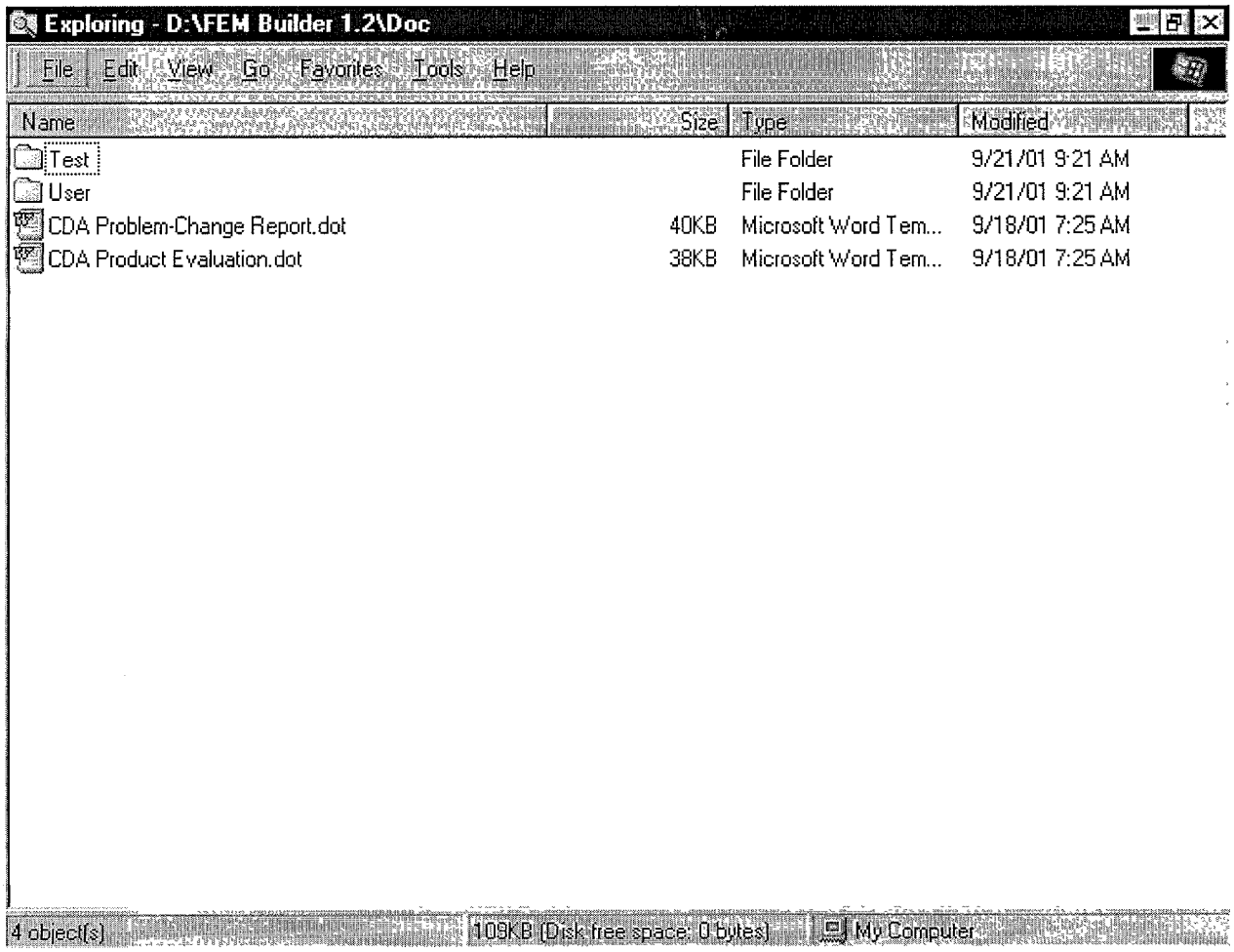
The following pages comprise a computer program listing appendix in the form of printouts of the file names and directories contained on the compact disk that comprises Appendix II of this specification. Included are directories and subdirectories, file and directory names, file types, file sizes in bytes, and creation dates.

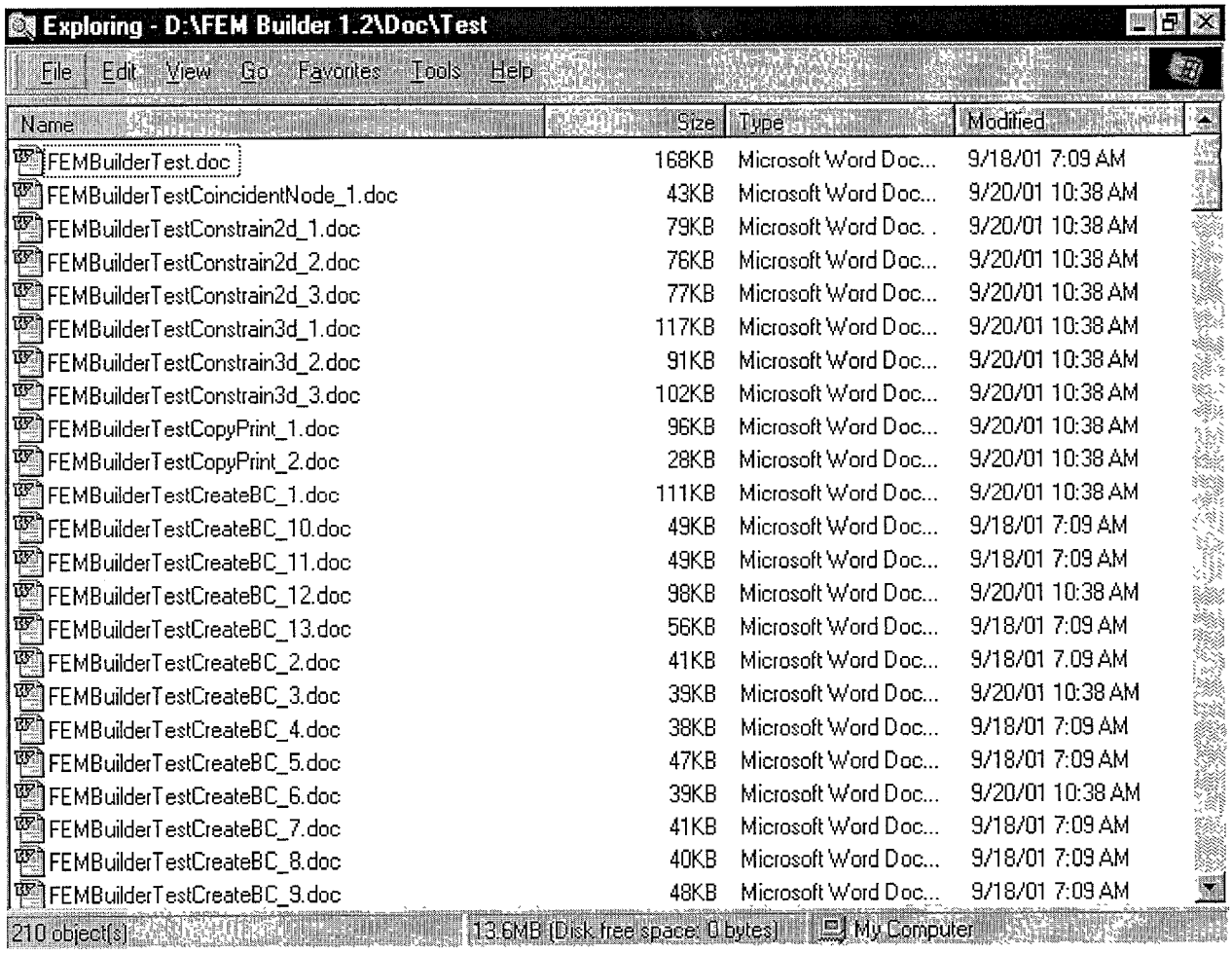

Exploring - D:\
File Edit View Go Favorites Tools Help
Name Size Type Modified
Ccm File Folder 9/21/01 9:21 AM
Cdca File Folder 9/21/01 9:21 AM
FEM Builder 1.2 File Folder 9/21/01 9:21 AM
Recess File Folder 9/21/01 9:21 AM
inst32i.ex 294KB EX_ File 11/19/97 4:05 PM
_isdel.exe 8KB Application 11/19/97 4:05 PM
_setup.dll 11KB Application Extension 11/19/97 4:08 PM
_sys1.cab 187KB Cabinet 9/21/01 8:29 AM
_user1.cab 50KB Cabinet 9/21/01 8:29 AM
Data.tag 1KB TAG File 9/21/01 8:29 AM
data1.cab 129KB Cabinet 9/21/01 8:29 AM
lang.dat 5KB Movie Clip 5/30/97 11:31 AM
layout.bin 1KB BIN File 9/21/01 8:29 AM
Mscreate.dir 0KB Macromedia Director. . 9/26/00 12:45 PM
os.dat 1KB Movie Clip 5/6/97 2 15 PM
py152.exe 4,911KB Application 7/16/99 10:54 AM
setup.bmp 81KB Bitmap Image 9/17/01 11:32 AM
Setup.exe 59KB Application 11/19/97 4:09 PM
Setup.ini 1KB Configuration Settings 9/21/01 8:29 AM
Setup.Ins 58KB Internet Communicati.. 9/21/01 8:28 AM
setup.lid 1KB LID File 9/21/01 8:29 AM
win32all-125.exe 2,953KB Application 2/22/00 10:51 AM
22 object(s) 8.54MB (Disk free space: 0 bytes) My Computer

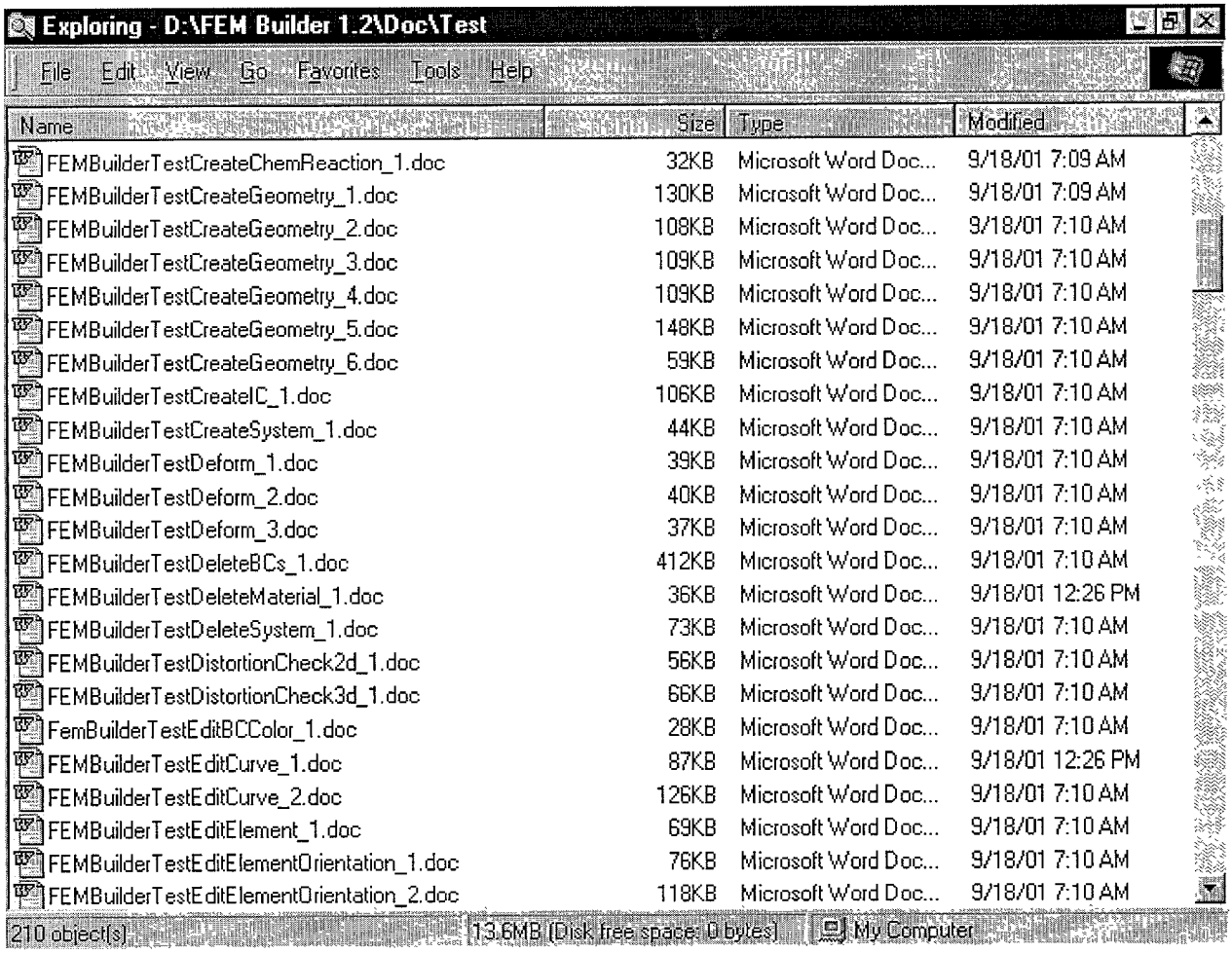

Exploring - D:\CCM
File Edit View Go Favorites Tools Help
Name Size Type Modified
CCM.bat 2KB MS-DOS Batch File 9/19/01 3:08 PM
CCM.exe 365KB Application 8/9/01 11:00 AM
CCMUser.doc 19KB Microsoft Word Doc... 9/19/01 9:35 AM
CCMW.exe 276KB Application 9/19/01 2:56 PM
4 object(s)
660KB (Disk free space: 0 bytes)
My Computer

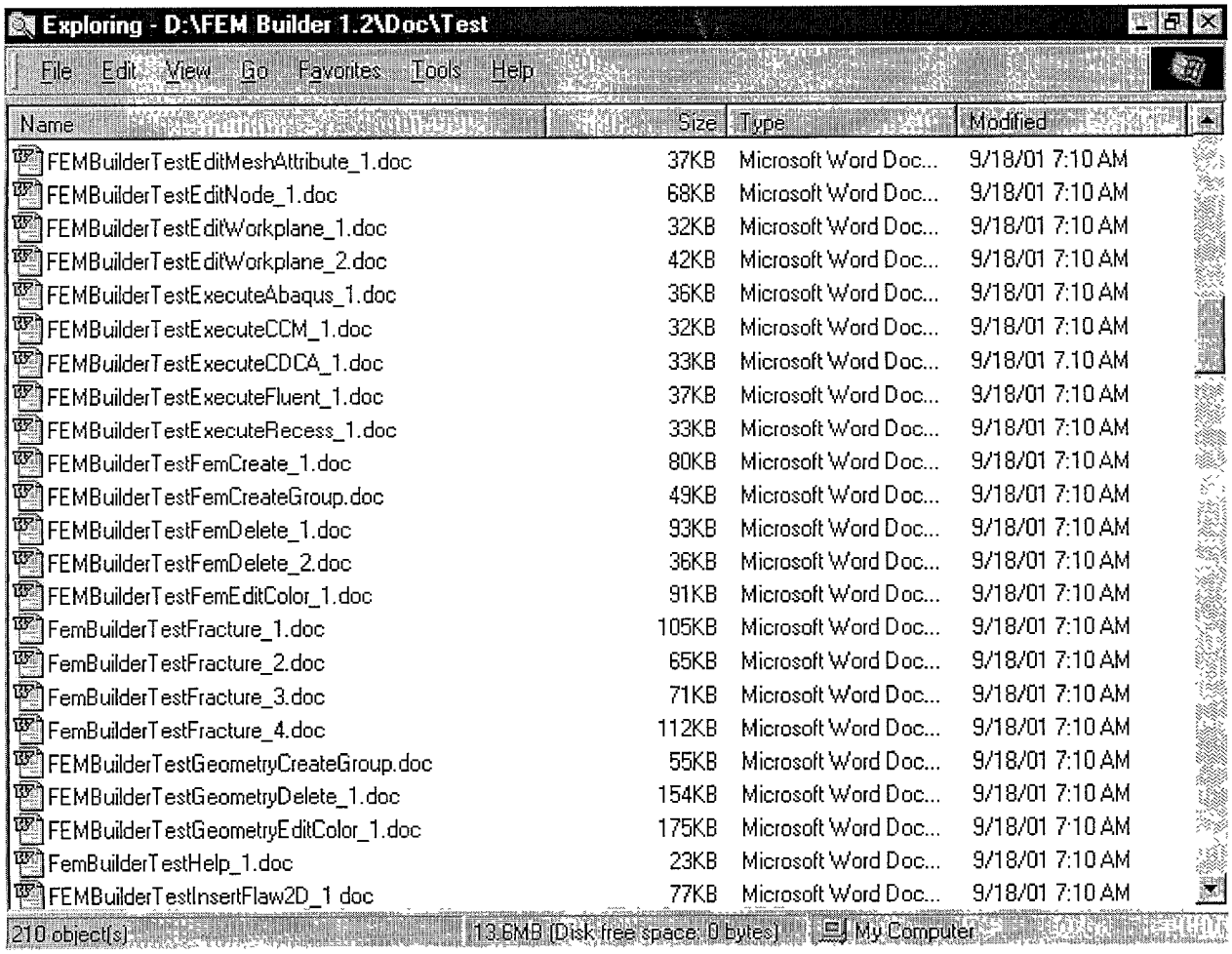

Exploring - D:\CDCA
File Edit View Go Favorites Tools Help
Name Size Type Modified
CDCA.bat 2KB MS-DOS Batch File 9/19/01 2:56 PM
CDCA.exe 545KB Application 7/25/01 1:20 PM
CDCAUser.doc 19KB Microsoft Word Doc... 9/19/01 9:34 AM
CDCAW.exe 72KB Application 9/19/01 2:56 PM
4 object(s)
636KB (Disk free space: 0 bytes)
My Computer

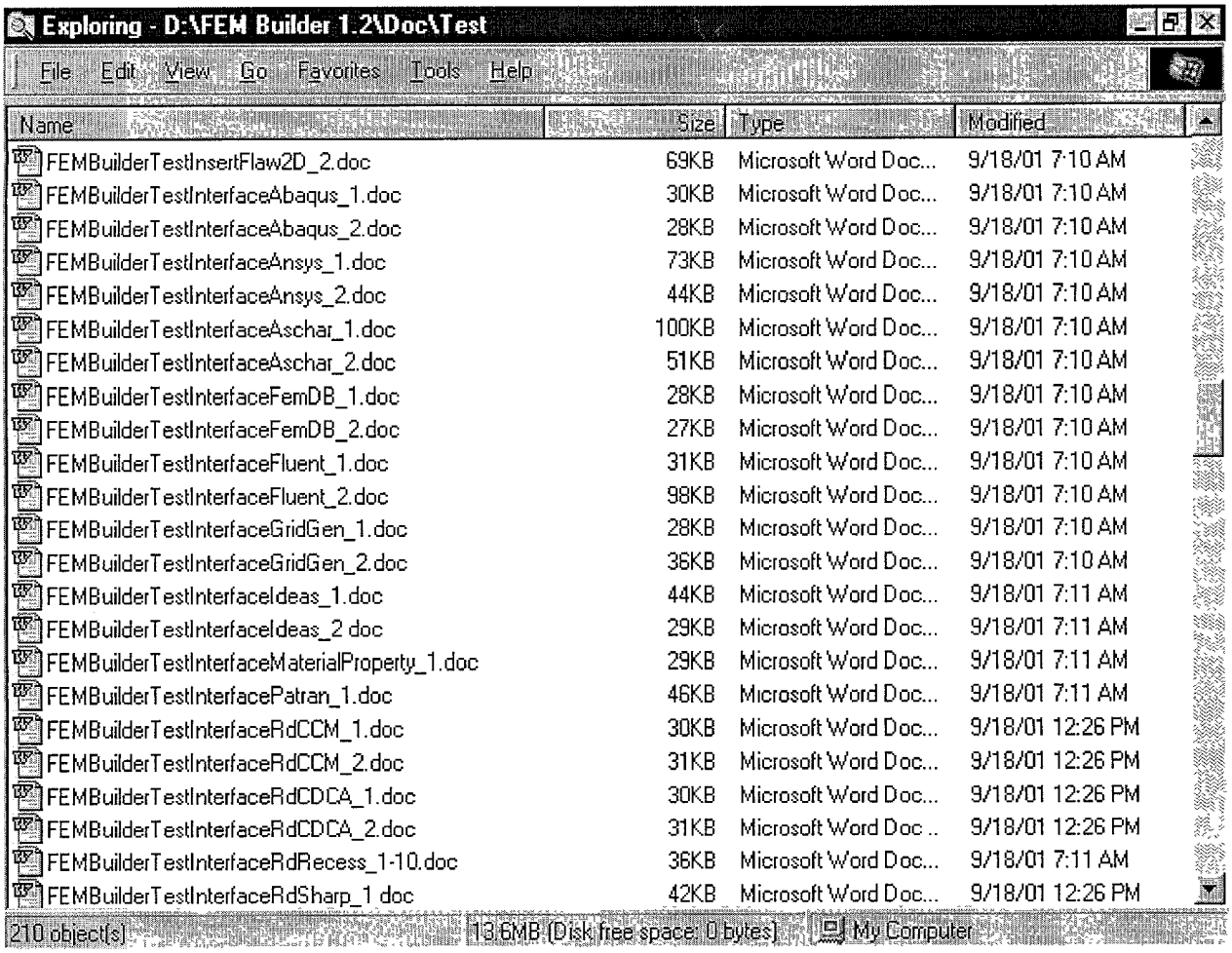

Exploring - D:\FEM Builder 1.2
File Edit View Go Favorites Tools Help
Name Size Type Modified
Doc File Folder 9/21/01 9:21 AM
Python File Folder 9/21/01 9:21 AM
Source File Folder 9/21/01 9:21 AM
Test File Folder 9/21/01 9:21 AM
error.wav 25KB Wave Sound 8/3/96 1:00 AM
Fem.exe 4,365KB Application 9/21/01 8:21 AM
FEM.pyd 1,652KB PYD File 9/21/01 8:18 AM
FemBuilderInstallationInstructions.doc 273KB Microsoft Word Doc... 9/19/01 3:17 PM
8 object(s)
6.18MB (Disk free space: 0 bytes)
My Computer

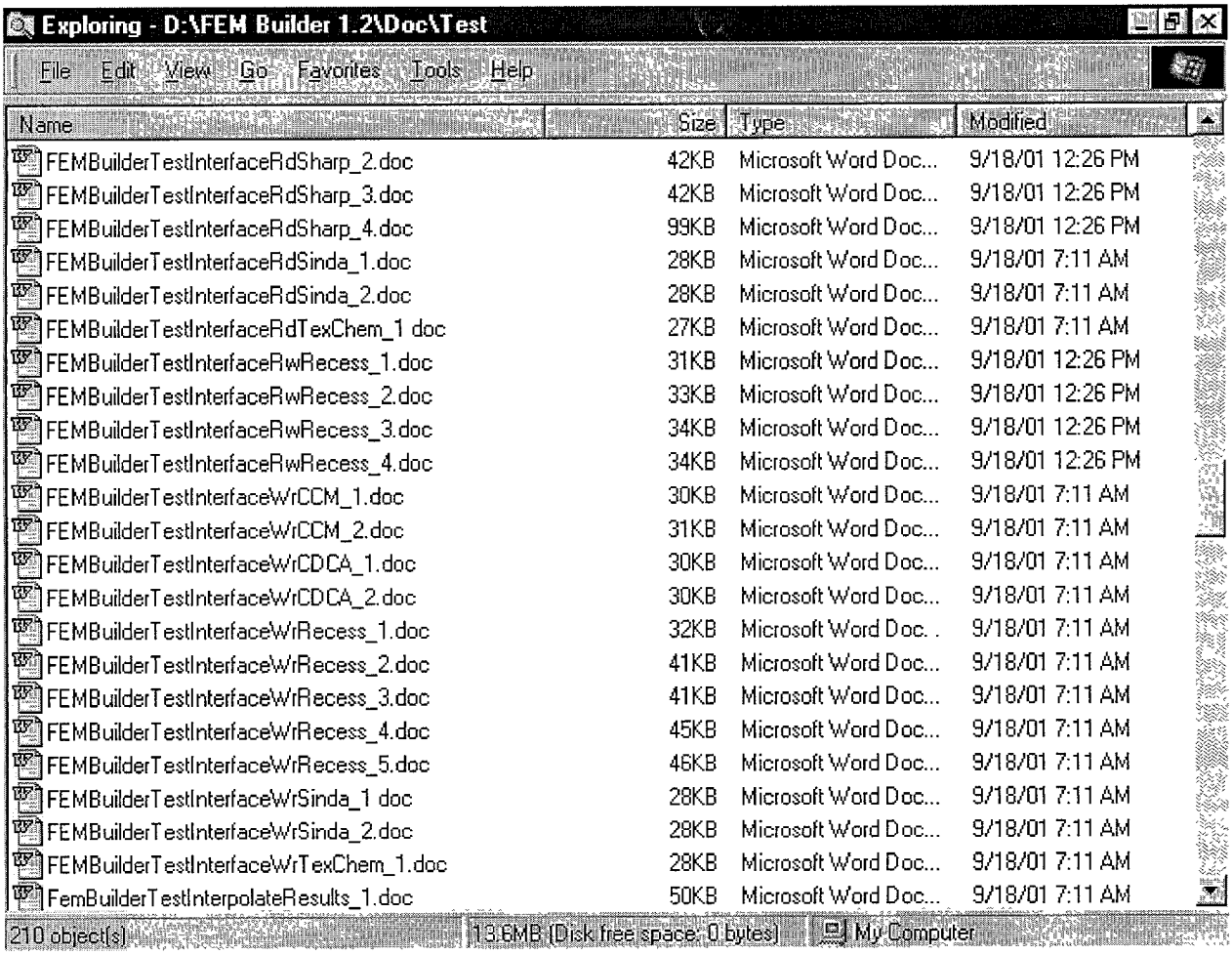

Exploring - D:\FEM Builder 1.2\Doc
File Edit View Go Favorites Tools Help
Name Size Type Modified
Test File Folder 9/21/01 9:21 AM
User File Folder 9/21/01 9:21 AM
CDA Problem-Change Report.dot 40KB Microsoft Word Tem... 9/18/01 7:25 AM
CDA Product Evaluation.dot 38KB Microsoft Word Tem... 9/18/01 7:25 AM
4 object(s)
109KB (Disk free space: 0 bytes)
My Computer

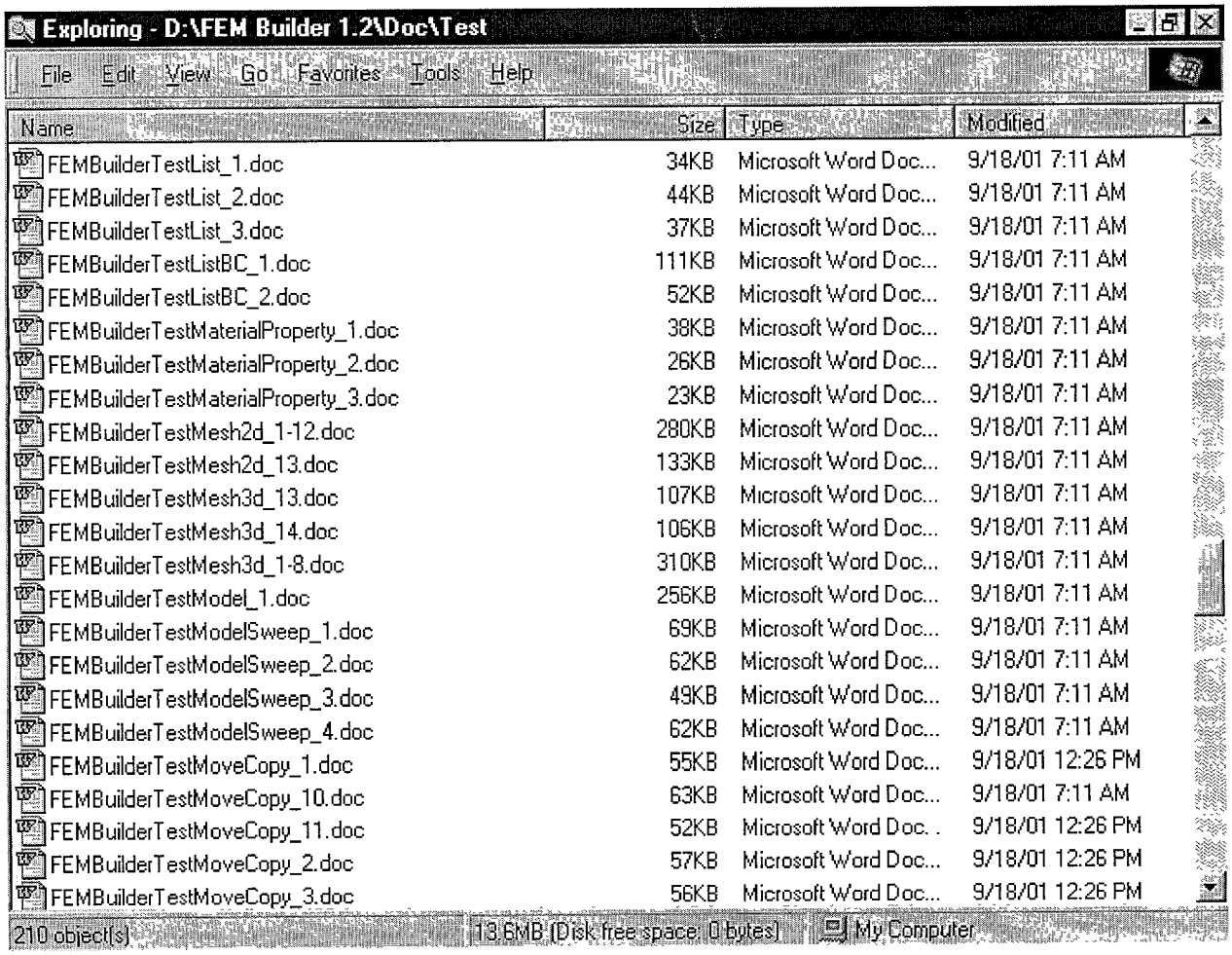

Exploring - D:\FEM Builder 1.2\Doc\Test
File Edit View Go Favorites Tools Help
Name Size Type Modified
FEMBuilderTest.doc 168KB Microsoft Word Doc... 9/18/01 7:09 AM
FEMBuilderTestCoincidentNode_1.doc 43KB Microsoft Word Doc... 9/20/01 10:38 AM
FEMBuilderTestConstrain2d_1.doc 79KB Microsoft Word Doc. . 9/20/01 10:38 AM
FEMBuilderTestConstrain2d_2.doc 76KB Microsoft Word Doc... 9/20/01 10:38 AM
FEMBuilderTestConstrain2d_3.doc 77KB Microsoft Word Doc... 9/20/01 10:38 AM
FEMBuilderTestConstrain3d_1.doc 117KB Microsoft Word Doc... 9/20/01 10:38 AM
FEMBuilderTestConstrain3d_2.doc 91KB Microsoft Word Doc... 9/20/01 10:38 AM
FEMBuilderTestConstrain3d_3.doc 102KB Microsoft Word Doc... 9/20/01 10:38 AM
FEMBuilderTestCopyPrint_1.doc 96KB Microsoft Word Doc... 9/20/01 10:38 AM
FEMBuilderTestCopyPrint_2.doc 28KB Microsoft Word Doc... 9/20/01 10:38 AM
FEMBuilderTestCreateBC_1.doc 111KB Microsoft Word Doc... 9/20/01 10:38 AM
FEMBuilderTestCreateBC_10.doc 49KB Microsoft Word Doc... 9/18/01 7:09 AM
FEMBuilderTestCreateBC_11.doc 49KB Microsoft Word Doc... 9/18/01 7:09 AM
FEMBuilderTestCreateBC_12.doc 98KB Microsoft Word Doc... 9/20/01 10:38 AM
FEMBuilderTestCreateBC_13.doc 56KB Microsoft Word Doc... 9/18/01 7:09 AM
FEMBuilderTestCreateBC_2.doc 41KB Microsoft Word Doc... 9/18/01 7:09 AM
FEMBuilderTestCreateBC_3.doc 39KB Microsoft Word Doc... 9/20/01 10:38 AM
FEMBuilderTestCreateBC_4.doc 38KB Microsoft Word Doc... 9/18/01 7:09 AM
FEMBuilderTestCreateBC_5.doc 47KB Microsoft Word Doc... 9/18/01 7:09 AM
FEMBuilderTestCreateBC_6.doc 39KB Microsoft Word Doc... 9/20/01 10:38 AM
FEMBuilderTestCreateBC_7.doc 41KB Microsoft Word Doc... 9/18/01 7:09 AM
FEMBuilderTestCreateBC_8.doc 40KB Microsoft Word Doc... 9/18/01 7:09 AM
FEMBuilderTestCreateBC_9.doc 48KB Microsoft Word Doc... 9/18/01 7:09 AM
210 object(s) 13.6MB (Disk free space: 0 bytes) My Computer

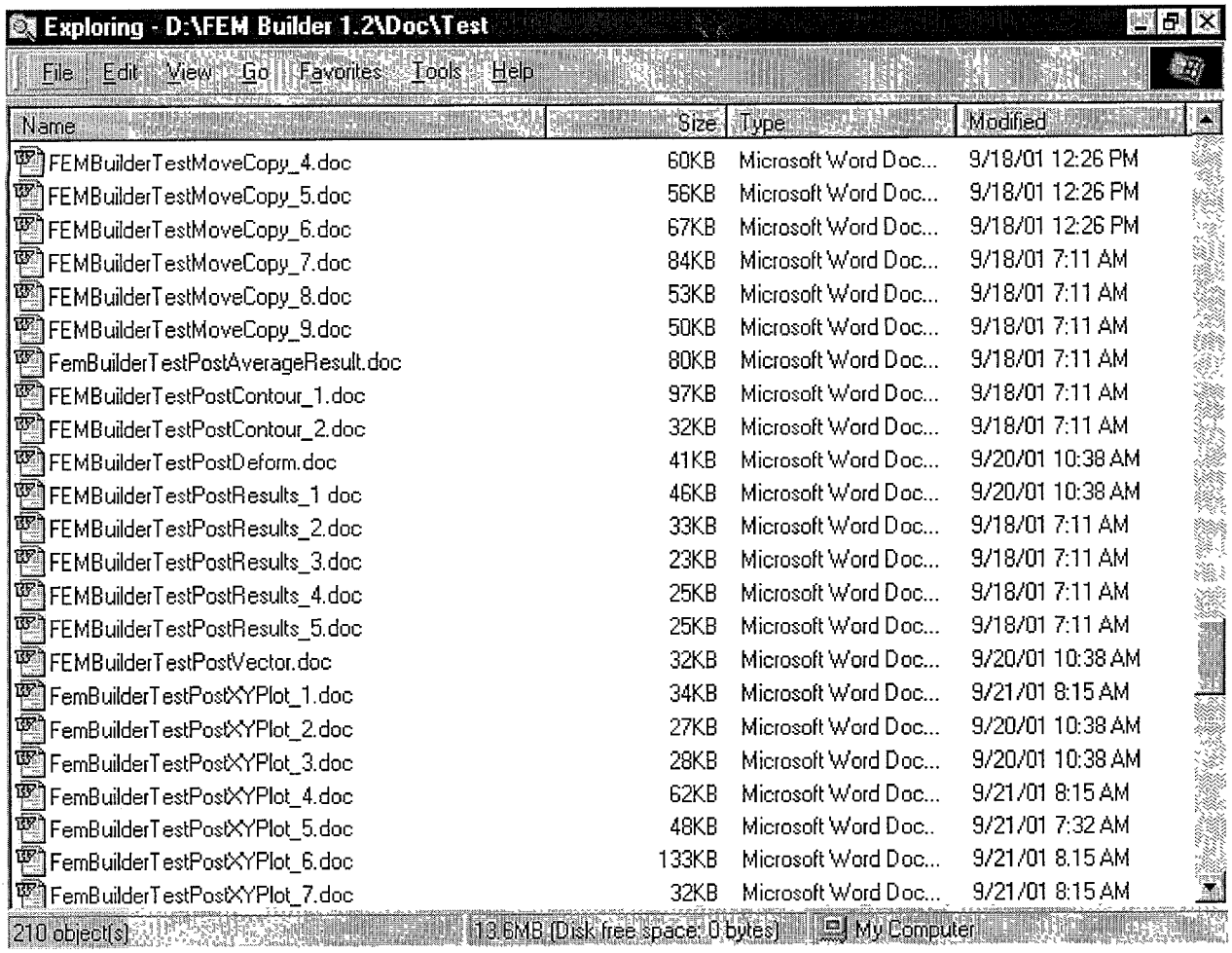

Exploring - D:\FEM Builder 1.2\Doc\Test
File Edit View Go Favorites Tools Help
Name Size Type Modified
FEMBuilderTestCreateChemReaction_1.doc 32KB Microsoft Word Doc... 9/18/01 7:09 AM
FEMBuilderTestCreateGeometry_1.doc 130KB Microsoft Word Doc... 9/18/01 7:09 AM
FEMBuilderTestCreateGeometry_2.doc 108KB Microsoft Word Doc... 9/18/01 7:10 AM
FEMBuilderTestCreateGeometry_3.doc 109KB Microsoft Word Doc... 9/18/01 7:10 AM
FEMBuilderTestCreateGeometry_4.doc 109KB Microsoft Word Doc... 9/18/01 7:10 AM
FEMBuilderTestCreateGeometry_5.doc 148KB Microsoft Word Doc... 9/18/01 7:10 AM
FEMBuilderTestCreateGeometry_6.doc 59KB Microsoft Word Doc... 9/18/01 7:10 AM
FEMBuilderTestCreateIC_1.doc 106KB Microsoft Word Doc... 9/18/01 7:10 AM
FEMBuilderTestCreateSystem_1.doc 44KB Microsoft Word Doc... 9/18/01 7:10 AM
FEMBuilderTestDeform_1.doc 39KB Microsoft Word Doc... 9/18/01 7:10 AM
FEMBuilderTestDeform_2.doc 40KB Microsoft Word Doc... 9/18/01 7:10 AM
FEMBuilderTestDeform_3.doc 37KB Microsoft Word Doc... 9/18/01 7:10 AM
FEMBuilderTestDeleteBCs_1.doc 412KB Microsoft Word Doc... 9/18/01 7:10 AM
FEMBuilderTestDeleteMaterial_1.doc 36KB Microsoft Word Doc... 9/18/01 12:26 PM
FEMBuilderTestDeleteSystem_1.doc 73KB Microsoft Word Doc... 9/18/01 7:10 AM
FEMBuilderTestDistortionCheck2d_1.doc 56KB Microsoft Word Doc... 9/18/01 7:10 AM
FEMBuilderTestDistortionCheck3d_1.doc 66KB Microsoft Word Doc... 9/18/01 7:10 AM
FemBuilderTestEditBCColor_1.doc 28KB Microsoft Word Doc... 9/18/01 7:10 AM
FEMBuilderTestEditCurve_1.doc 87KB Microsoft Word Doc... 9/18/01 12:26 PM
FEMBuilderTestEditCurve_2.doc 126KB Microsoft Word Doc... 9/18/01 7:10 AM
FEMBuilderTestEditElement_1.doc 69KB Microsoft Word Doc... 9/18/01 7:10 AM
FEMBuilderTestEditElementOrientation_1.doc 76KB Microsoft Word Doc... 9/18/01 7:10 AM
FEMBuilderTestEditElementOrientation_2.doc 118KB Microsoft Word Doc... 9/18/01 7:10 AM
210 object(s) 13.6MB (Disk free space: 0 bytes) My Computer

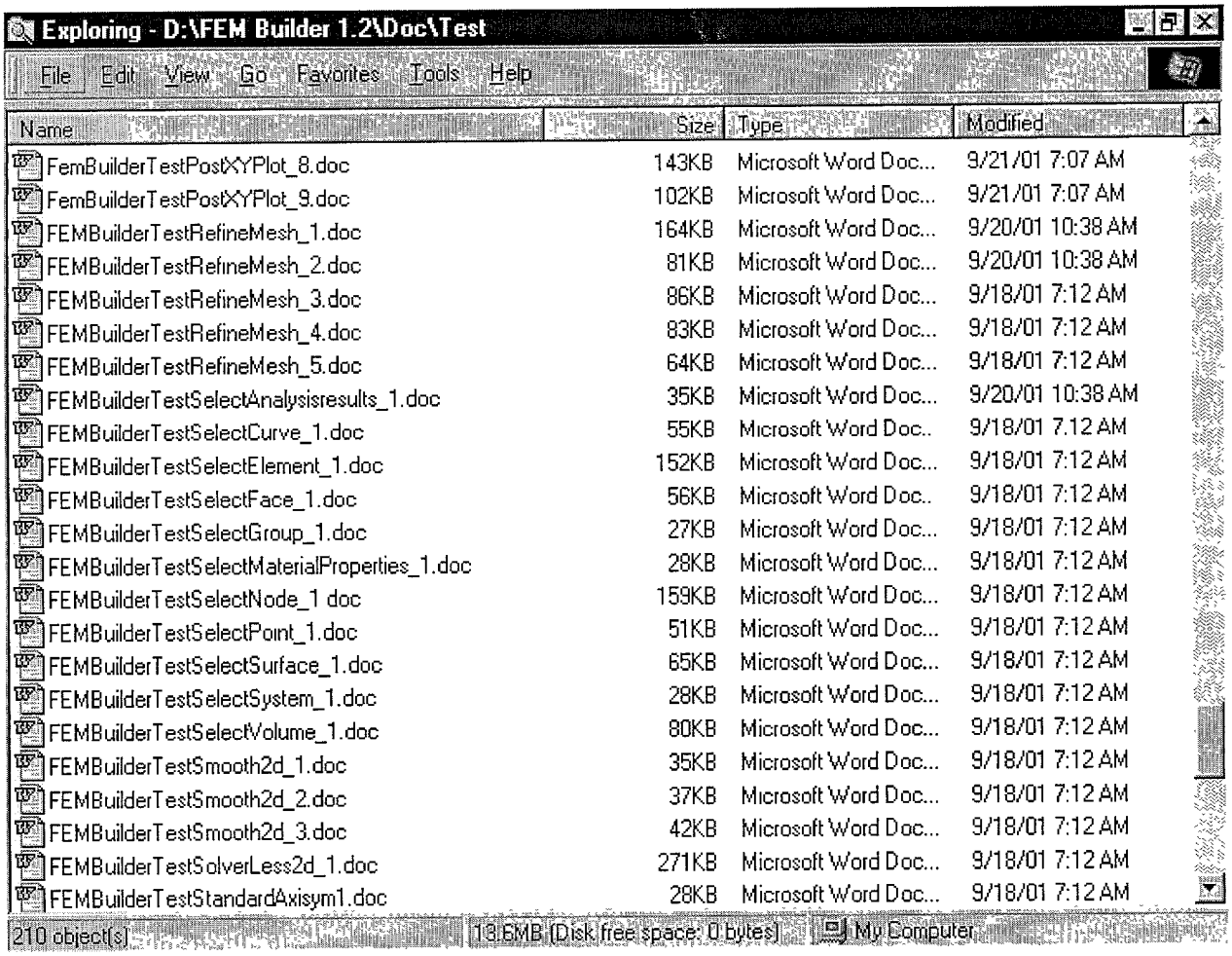

Exploring - D:\FEM Builder 1.2\Doc\Test
File Edit View Go Favorites Tools Help
Name Size Type Modified
FEMBuilderTestEditMeshAttribute_1.doc 37KB Microsoft Word Doc... 9/18/01 7:10 AM
FEMBuilderTestEditNode_1.doc 68KB Microsoft Word Doc... 9/18/01 7:10 AM
FEMBuilderTestEditWorkplane_1.doc 32KB Microsoft Word Doc... 9/18/01 7:10 AM
FEMBuilderTestEditWorkplane_2.doc 42KB Microsoft Word Doc... 9/18/01 7:10 AM
FEMBuilderTestExecuteAbaqus_1.doc 36KB Microsoft Word Doc... 9/18/01 7:10 AM
FEMBuilderTestExecuteCCM_1.doc 32KB Microsoft Word Doc... 9/18/01 7:10 AM
FEMBuilderTestExecuteCDCA_1.doc 33KB Microsoft Word Doc... 9/18/01 7:10 AM
FEMBuilderTestExecuteFluent_1.doc 37KB Microsoft Word Doc... 9/18/01 7:10 AM
FEMBuilderTestExecuteRecess_1.doc 33KB Microsoft Word Doc... 9/18/01 7:10 AM
FEMBuilderTestFemCreate_1.doc 80KB Microsoft Word Doc... 9/18/01 7:10 AM
FEMBuilderTestFemCreateGroup.doc 49KB Microsoft Word Doc... 9/18/01 7:10 AM
FEMBuilderTestFemDelete_1.doc 93KB Microsoft Word Doc... 9/18/01 7:10 AM
FEMBuilderTestFemDelete_2.doc 36KB Microsoft Word Doc... 9/18/01 7:10 AM
FEMBuilderTestFemEditColor_1.doc 91KB Microsoft Word Doc... 9/18/01 7:10 AM
FemBuilderTestFracture_1.doc 105KB Microsoft Word Doc... 9/18/01 7:10 AM
FemBuilderTestFracture_2.doc 65KB Microsoft Word Doc... 9/18/01 7:10 AM
FemBuilderTestFracture_3.doc 71KB Microsoft Word Doc... 9/18/01 7:10 AM
FemBuilderTestFracture_4.doc 112KB Microsoft Word Doc... 9/18/01 7:10 AM
FEMBuilderTestGeometryCreateGroup.doc 55KB Microsoft Word Doc... 9/18/01 7:10 AM
FEMBuilderTestGeometryDelete_1.doc 154KB Microsoft Word Doc... 9/18/01 7:10 AM
FEMBuilderTestGeometryEditColor_1.doc 175KB Microsoft Word Doc... 9/18/01 7:10 AM
FemBuilderTestHelp_1.doc 23KB Microsoft Word Doc... 9/18/01 7:10 AM
FEMBuilderTestInsertFlaw2D_1.doc 77KB Microsoft Word Doc... 9/18/01 7:10 AM
210 object(s) 13.8MB (Disk free space: 0 bytes) My Computer

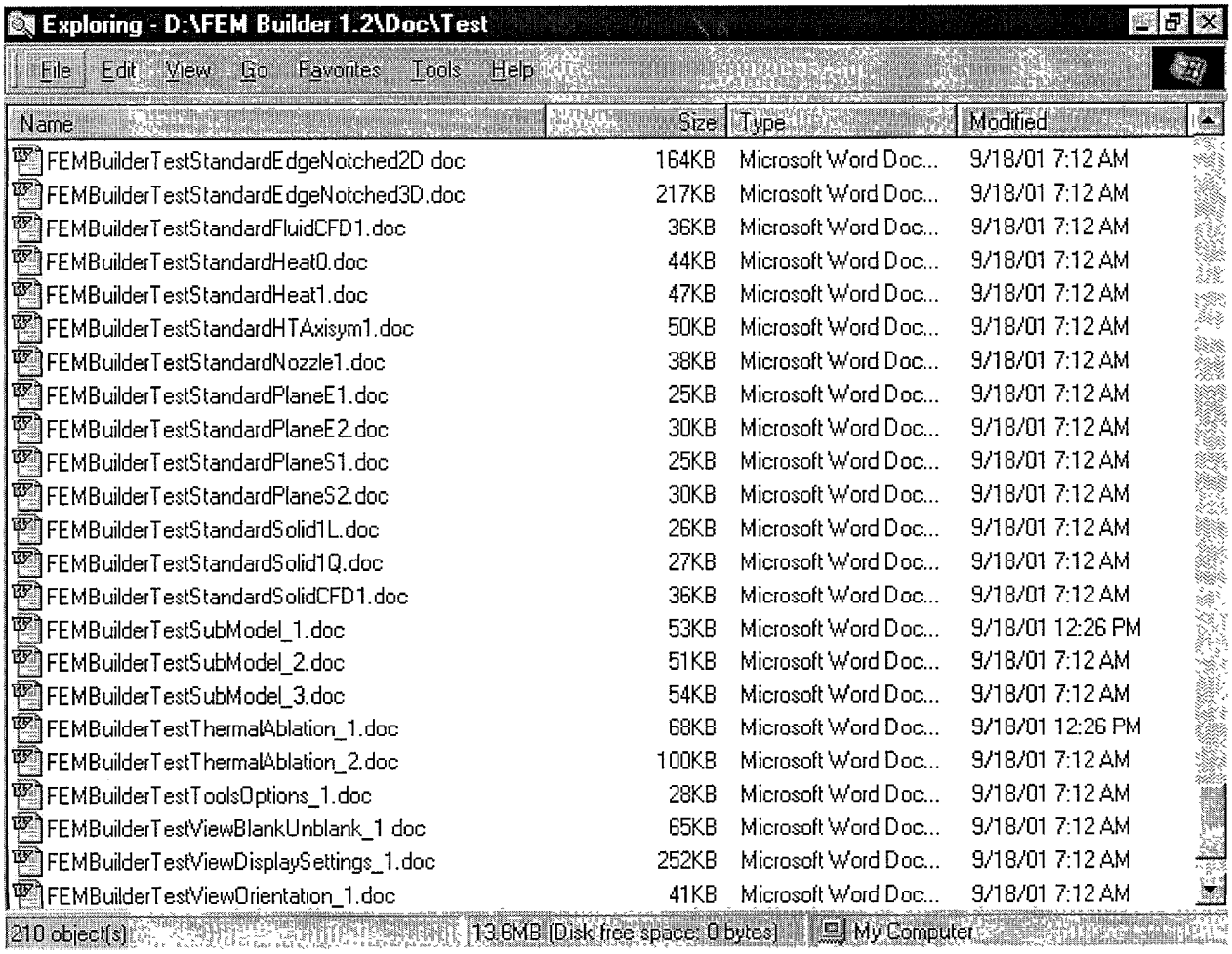

Exploring - D:\FEM Builder 1.2\Doc\Test
File Edit View Go Favorites Tools Help
Name Size Type Modified
FEMBuilderTestInsertFlaw2D_2.doc 69KB Microsoft Word Doc... 9/18/01 7:10 AM
FEMBuilderTestInterfaceAbaqus_1.doc 30KB Microsoft Word Doc... 9/18/01 7:10 AM
FEMBuilderTestInterfaceAbaqus_2.doc 28KB Microsoft Word Doc... 9/18/01 7:10 AM
FEMBuilderTestInterfaceAnsys_1.doc 73KB Microsoft Word Doc... 9/18/01 7:10 AM
FEMBuilderTestInterfaceAnsys_2.doc 44KB Microsoft Word Doc... 9/18/01 7:10 AM
FEMBuilderTestInterfaceAschar_1.doc 100KB Microsoft Word Doc... 9/18/01 7:10 AM
FEMBuilderTestInterfaceAschar_2.doc 51KB Microsoft Word Doc... 9/18/01 7:10 AM
FEMBuilderTestInterfaceFemDB_1.doc 28KB Microsoft Word Doc... 9/18/01 7:10 AM
FEMBuilderTestInterfaceFemDB_2.doc 27KB Microsoft Word Doc... 9/18/01 7:10 AM
FEMBuilderTestInterfaceFluent_1.doc 31KB Microsoft Word Doc... 9/18/01 7:10 AM
FEMBuilderTestInterfaceFluent_2.doc 98KB Microsoft Word Doc... 9/18/01 7:10 AM
FEMBuilderTestInterfaceGridGen_1.doc 28KB Microsoft Word Doc... 9/18/01 7:10 AM
FEMBuilderTestInterfaceGridGen_2.doc 36KB Microsoft Word Doc... 9/18/01 7:10 AM
FEMBuilderTestInterfaceIdeas_1.doc 44KB Microsoft Word Doc... 9/18/01 7:11 AM
FEMBuilderTestInterfaceIdeas_2.doc 29KB Microsoft Word Doc... 9/18/01 7:11 AM
FEMBuilderTestInterfaceMaterialProperty_1.doc 29KB Microsoft Word Doc... 9/18/01 7:11 AM
FEMBuilderTestInterfacePatran_1.doc 46KB Microsoft Word Doc... 9/18/01 7:11 AM
FEMBuilderTestInterfaceRdCCM_1.doc 30KB Microsoft Word Doc... 9/18/01 12:26 PM
FEMBuilderTestInterfaceRdCCM_2.doc 31KB Microsoft Word Doc... 9/18/01 12:26 PM
FEMBuilderTestInterfaceRdCDCA_1.doc 30KB Microsoft Word Doc... 9/18/01 12:26 PM
FEMBuilderTestInterfaceRdCDCA_2.doc 31KB Microsoft Word Doc .. 9/18/01 12:26 PM
FEMBuilderTestInterfaceRdRecess_1-10.doc 36KB Microsoft Word Doc... 9/18/01 7:11 AM
FEMBuilderTestInterfaceRdSharp_1.doc 42KB Microsoft Word Doc... 9/18/01 12:26 PM
210 object(s) 13.6MB (Disk free space: 0 bytes) My Computer

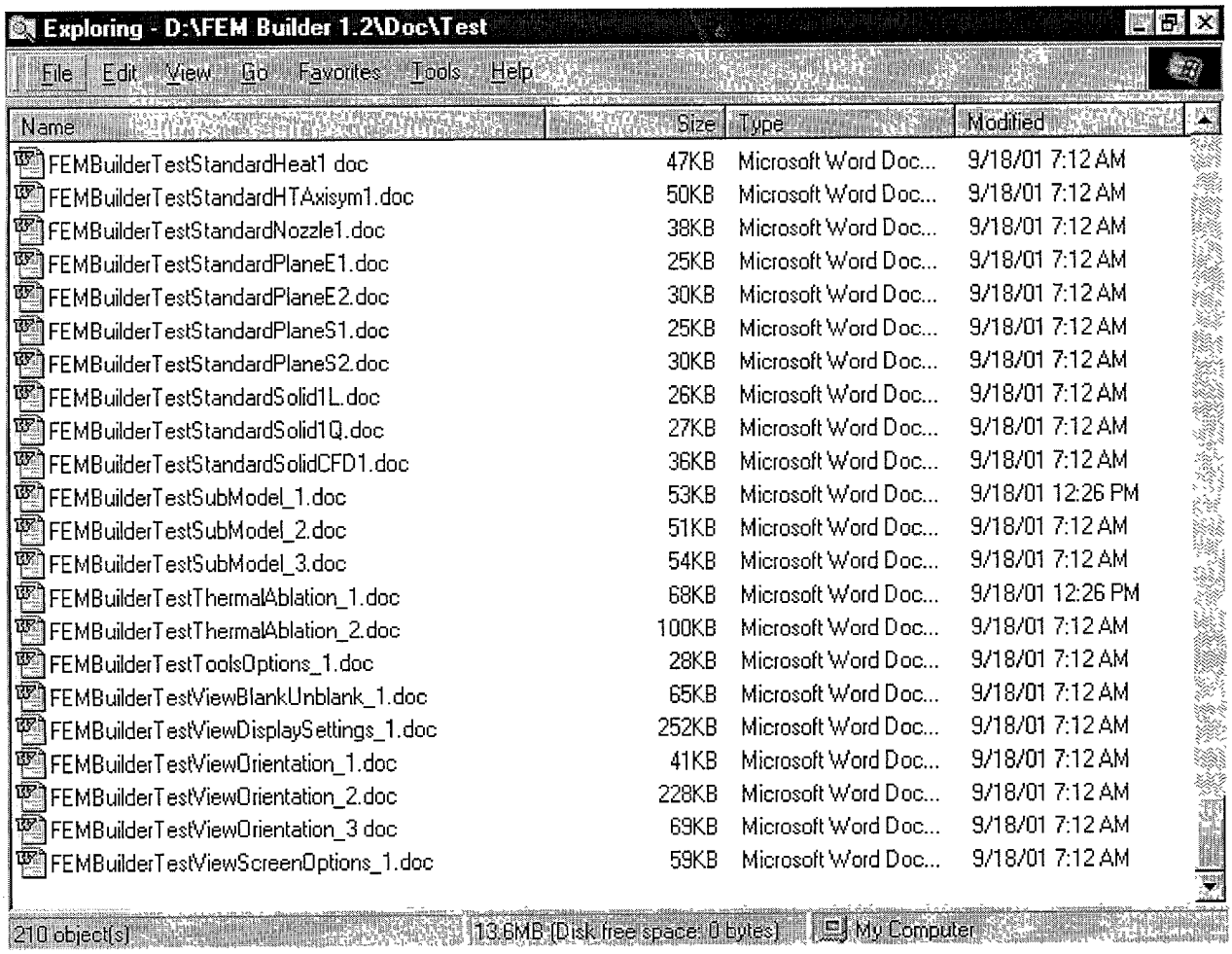

Exploring - D:\FEM Builder 1.2\Doc\Test
File Edit View Go Favorites Tools Help
Name Size Type Modified
FEMBuilderTestInterfaceRdSharp_2.doc 42KB Microsoft Word Doc... 9/18/01 12:26 PM
FEMBuilderTestInterfaceRdSharp_3.doc 42KB Microsoft Word Doc... 9/18/01 12:26 PM
FEMBuilderTestInterfaceRdSharp_4.doc 99KB Microsoft Word Doc... 9/18/01 12:26 PM
FEMBuilderTestInterfaceRdSinda_1.doc 28KB Microsoft Word Doc... 9/18/01 7:11 AM
FEMBuilderTestInterfaceRdSinda_2.doc 28KB Microsoft Word Doc... 9/18/01 7:11 AM
FEMBuilderTestInterfaceRdTexChem_1 doc 27KB Microsoft Word Doc... 9/18/01 7:11 AM
FEMBuilderTestInterfaceRwRecess_1.doc 31KB Microsoft Word Doc... 9/18/01 12:26 PM
FEMBuilderTestInterfaceRwRecess_2.doc 33KB Microsoft Word Doc... 9/18/01 12:26 PM
FEMBuilderTestInterfaceRwRecess_3.doc 34KB Microsoft Word Doc... 9/18/01 12:26 PM
FEMBuilderTestInterfaceRwRecess_4.doc 34KB Microsoft Word Doc... 9/18/01 12:26 PM
FEMBuilderTestInterfaceWrCCM_1.doc 30KB Microsoft Word Doc... 9/18/01 7:11 AM
FEMBuilderTestInterfaceWrCCM_2.doc 31KB Microsoft Word Doc... 9/18/01 7:11 AM
FEMBuilderTestInterfaceWrCDCA_1.doc 30KB Microsoft Word Doc... 9/18/01 7:11 AM
FEMBuilderTestInterfaceWrCDCA_2.doc 30KB Microsoft Word Doc... 9/18/01 7:11 AM
FEMBuilderTestInterfaceWrRecess_1.doc 32KB Microsoft Word Doc. . 9/18/01 7:11 AM
FEMBuilderTestInterfaceWrRecess_2.doc 41KB Microsoft Word Doc... 9/18/01 7:11 AM
FEMBuilderTestInterfaceWrRecess_3.doc 41KB Microsoft Word Doc... 9/18/01 7:11 AM
FEMBuilderTestInterfaceWrRecess_4.doc 45KB Microsoft Word Doc... 9/18/01 7:11 AM
FEMBuilderTestInterfaceWrRecess_5.doc 46KB Microsoft Word Doc... 9/18/01 7:11 AM
FEMBuilderTestInterfaceWrSinda_1 doc 28KB Microsoft Word Doc... 9/18/01 7:11 AM
FEMBuilderTestInterfaceWrSinda_2.doc 28KB Microsoft Word Doc... 9/18/01 7:11 AM
FEMBuilderTestInterfaceWrTexChem_1.doc 28KB Microsoft Word Doc... 9/18/01 7:11 AM
FemBuilderTestInterpolateResults_1.doc 50KB Microsoft Word Doc... 9/18/01 7:11 AM
210 object(s) 13.6MB (Disk free space: 0 bytes) My Computer

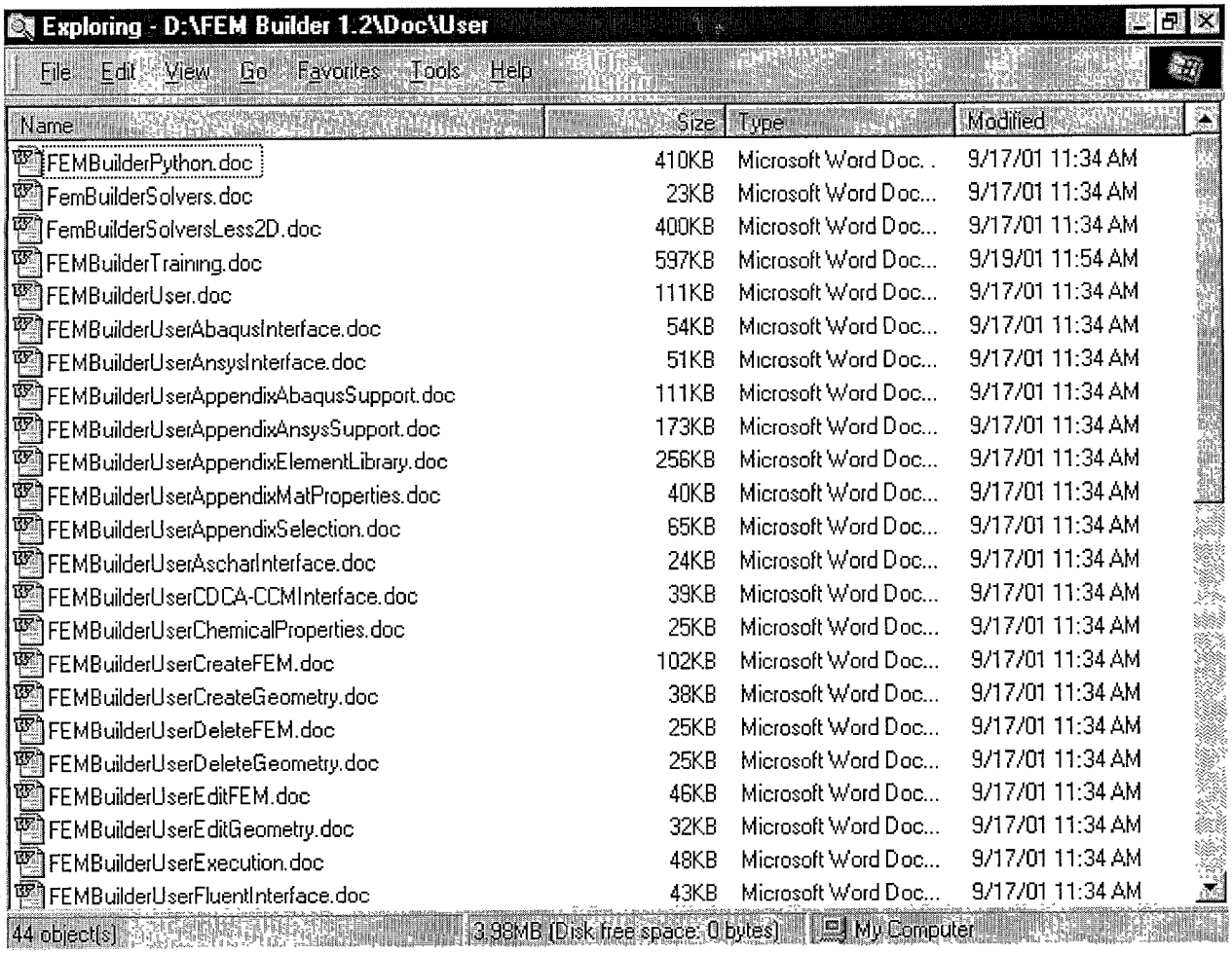

Exploring - D:\FEM Builder 1.2\Doc\Test
File Edit View Go Favorites Tools Help
Name Size Type Modified
FEMBuilderTestList_1.doc 34KB Microsoft Word Doc... 9/18/01 7:11 AM
FEMBuilderTestList_2.doc 44KB Microsoft Word Doc... 9/18/01 7:11 AM
FEMBuilderTestList_3.doc 37KB Microsoft Word Doc... 9/18/01 7:11 AM
FEMBuilderTestListBC_1.doc 111KB Microsoft Word Doc... 9/18/01 7:11 AM
FEMBuilderTestListBC_2.doc 52KB Microsoft Word Doc... 9/18/01 7:11 AM
FEMBuilderTestMaterialProperty_1.doc 38KB Microsoft Word Doc... 9/18/01 7:11 AM
FEMBuilderTestMaterialProperty_2.doc 26KB Microsoft Word Doc... 9/18/01 7:11 AM
FEMBuilderTestMaterialProperty_3.doc 23KB Microsoft Word Doc... 9/18/01 7:11 AM
FEMBuilderTestMesh2d_1-12.doc 280KB Microsoft Word Doc... 9/18/01 7:11 AM
FEMBuilderTestMesh2d_13.doc 133KB Microsoft Word Doc... 9/18/01 7:11 AM
FEMBuilderTestMesh3d_13.doc 107KB Microsoft Word Doc... 9/18/01 7:11 AM
FEMBuilderTestMesh3d_14.doc 106KB Microsoft Word Doc... 9/18/01 7:11 AM
FEMBuilderTestMesh3d_1-8.doc 310KB Microsoft Word Doc... 9/18/01 7:11 AM
FEMBuilderTestModel_1.doc 256KB Microsoft Word Doc... 9/18/01 7:11 AM
FEMBuilderTestModelSweep_1.doc 69KB Microsoft Word Doc... 9/18/01 7:11 AM
FEMBuilderTestModelSweep_2.doc 62KB Microsoft Word Doc... 9/18/01 7:11 AM
FEMBuilderTestModelSweep_3.doc 49KB Microsoft Word Doc... 9/18/01 7:11 AM
FEMBuilderTestModelSweep_4.doc 62KB Microsoft Word Doc... 9/18/01 7:11 AM
FEMBuilderTestMoveCopy_1.doc 55KB Microsoft Word Doc... 9/18/01 12:26 PM
FEMBuilderTestMoveCopy_10.doc 63KB Microsoft Word Doc... 9/18/01 7:11 AM
FEMBuilderTestMoveCopy_11.doc 52KB Microsoft Word Doc. . 9/18/01 12:26 PM
FEMBuilderTestMoveCopy_2.doc 57KB Microsoft Word Doc... 9/18/01 12:26 PM
FEMBuilderTestMoveCopy_3.doc 56KB Microsoft Word Doc... 9/18/01 12:26 PM
210 object(s) 13.6MB (Disk free space: 0 bytes) My Computer

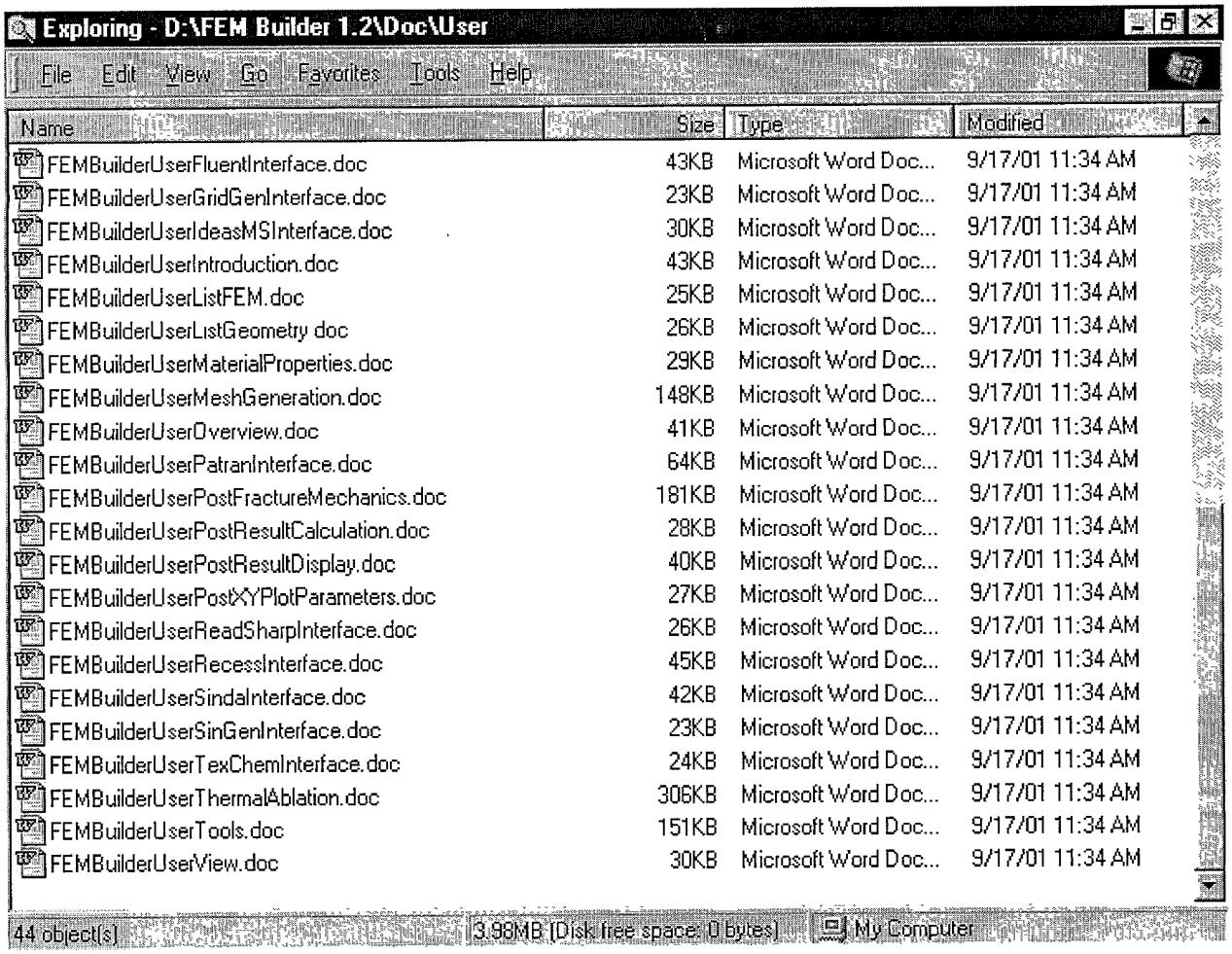

Exploring - D:\FEM Builder 1.2\Doc\Test
File Edit View Go Favorites Tools Help
Name Size Type Modified
FEMBuilderTestMoveCopy_4.doc 60KB Microsoft Word Doc... 9/18/01 12:26 PM
FEMBuilderTestMoveCopy_5.doc 56KB Microsoft Word Doc... 9/18/01 12:26 PM
FEMBuilderTestMoveCopy_6.doc 67KB Microsoft Word Doc... 9/18/01 12:26 PM
FEMBuilderTestMoveCopy_7.doc 84KB Microsoft Word Doc... 9/18/01 7:11 AM
FEMBuilderTestMoveCopy_8.doc 53KB Microsoft Word Doc... 9/18/01 7:11 AM
FEMBuilderTestMoveCopy_9.doc 50KB Microsoft Word Doc... 9/18/01 7:11 AM
FemBuilderTestPostAverageResult.doc 80KB Microsoft Word Doc... 9/18/01 7:11 AM
FEMBuilderTestPostContour_1.doc 97KB Microsoft Word Doc... 9/18/01 7:11 AM
FEMBuilderTestPostContour_2.doc 32KB Microsoft Word Doc... 9/18/01 7:11 AM
FEMBuilderTestPostDeform.doc 41KB Microsoft Word Doc... 9/20/01 10:38 AM
FEMBuilderTestPostResults_1 doc 46KB Microsoft Word Doc... 9/20/01 10:38 AM
FEMBuilderTestPostResults_2.doc 33KB Microsoft Word Doc... 9/18/01 7:11 AM
FEMBuilderTestPostResults_3.doc 23KB Microsoft Word Doc... 9/18/01 7:11 AM
FEMBuilderTestPostResults_4.doc 25KB Microsoft Word Doc... 9/18/01 7:11 AM
FEMBuilderTestPostResults_5.doc 25KB Microsoft Word Doc... 9/18/01 7:11 AM
FEMBuilderTestPostVector.doc 32KB Microsoft Word Doc... 9/20/01 10:38 AM
FemBuilderTestPostXYPlot_1.doc 34KB Microsoft Word Doc... 9/21/01 8:15 AM
FemBuilderTestPostXYPlot_2.doc 27KB Microsoft Word Doc... 9/20/01 10:38 AM
FemBuilderTestPostXYPlot_3.doc 28KB Microsoft Word Doc... 9/20/01 10:38 AM
FemBuilderTestPostXYPlot_4.doc 62KB Microsoft Word Doc... 9/21/01 8:15 AM
FemBuilderTestPostXYPlot_5.doc 48KB Microsoft Word Doc.. 9/21/01 7:32 AM
FemBuilderTestPostXYPlot_6.doc 133KB Microsoft Word Doc... 9/21/01 8.15 AM
FemBuilderTestPostXYPlot_7.doc 32KB Microsoft Word Doc... 9/21/01 8:15 AM
210 object(s) 18.6MB (Disk free space: 0 bytes) My Computer

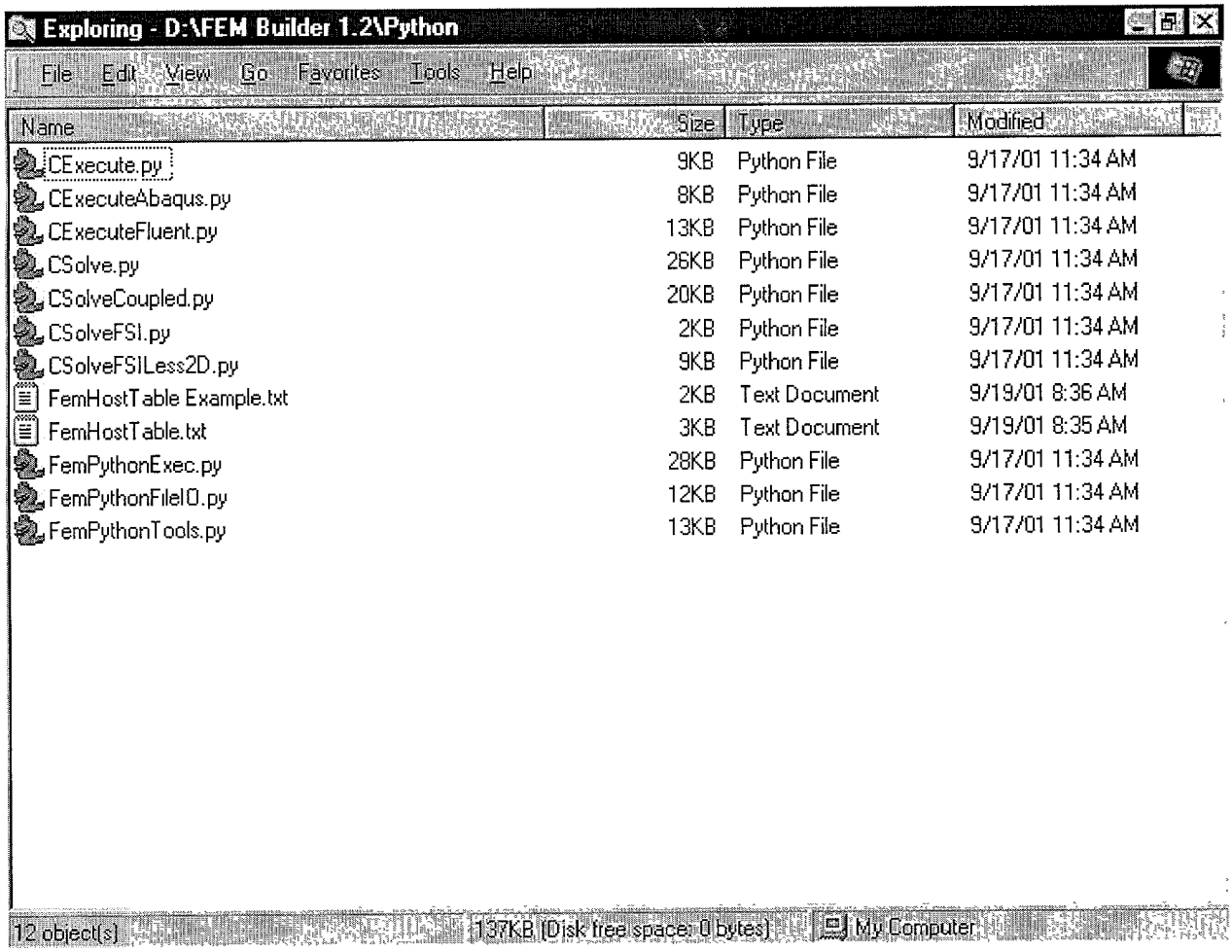

Exploring - D:\FEM Builder 1.2\Doc\Test
File Edit View Go Favorites Tools Help
Name Size Type Modified
FemBuilderTestPostXYPlot_8.doc 143KB Microsoft Word Doc... 9/21/01 7:07 AM
FemBuilderTestPostXYPlot_9.doc 102KB Microsoft Word Doc... 9/21/01 7:07 AM
FEMBuilderTestRefineMesh_1.doc 164KB Microsoft Word Doc... 9/20/01 10:38 AM
FEMBuilderTestRefineMesh_2.doc 81KB Microsoft Word Doc... 9/20/01 10:38 AM
FEMBuilderTestRefineMesh_3.doc 86KB Microsoft Word Doc... 9/18/01 7:12 AM
FEMBuilderTestRefineMesh_4.doc 83KB Microsoft Word Doc... 9/18/01 7:12 AM
FEMBuilderTestRefineMesh_5.doc 64KB Microsoft Word Doc... 9/18/01 7:12 AM
FEMBuilderTestSelectAnalysisresults_1.doc 35KB Microsoft Word Doc... 9/20/01 10:38 AM
FEMBuilderTestSelectCurve_1.doc 55KB Microsoft Word Doc.. 9/18/01 7:12 AM
FEMBuilderTestSelectElement_1.doc 152KB Microsoft Word Doc... 9/18/01 7:12 AM
FEMBuilderTestSelectFace_1.doc 56KB Microsoft Word Doc.. 9/18/01 7:12 AM
FEMBuilderTestSelectGroup_1.doc 27KB Microsoft Word Doc... 9/18/01 7:12 AM
FEMBuilderTestSelectMaterialProperties_1.doc 28KB Microsoft Word Doc... 9/18/01 7:12 AM
FEMBuilderTestSelectNode_1 doc 159KB Microsoft Word Doc... 9/18/01 7:12 AM
FEMBuilderTestSelectPoint_1.doc 51KB Microsoft Word Doc... 9/18/01 7:12 AM
FEMBuilderTestSelectSurface_1.doc 65KB Microsoft Word Doc... 9/18/01 7:12 AM
FEMBuilderTestSelectSystem_1.doc 28KB Microsoft Word Doc... 9/18/01 7:12 AM
FEMBuilderTestSelectVolume_1.doc 80KB Microsoft Word Doc... 9/18/01 7:12 AM
FEMBuilderTestSmooth2d_1.doc 35KB Microsoft Word Doc... 9/18/01 7:12 AM
FEMBuilderTestSmooth2d_2.doc 37KB Microsoft Word Doc... 9/18/01 7:12 AM
FEMBuilderTestSmooth2d_3.doc 42KB Microsoft Word Doc... 9/18/01 7:12 AM
FEMBuilderTestSolverLess2d_1.doc 271KB Microsoft Word Doc... 9/18/01 7:12 AM
FEMBuilderTestStandardAxisym1.doc 28KB Microsoft Word Doc... 9/18/01 7:12 AM
210 object(s) 13.6MB (Disk free space: 0 bytes) My Computer

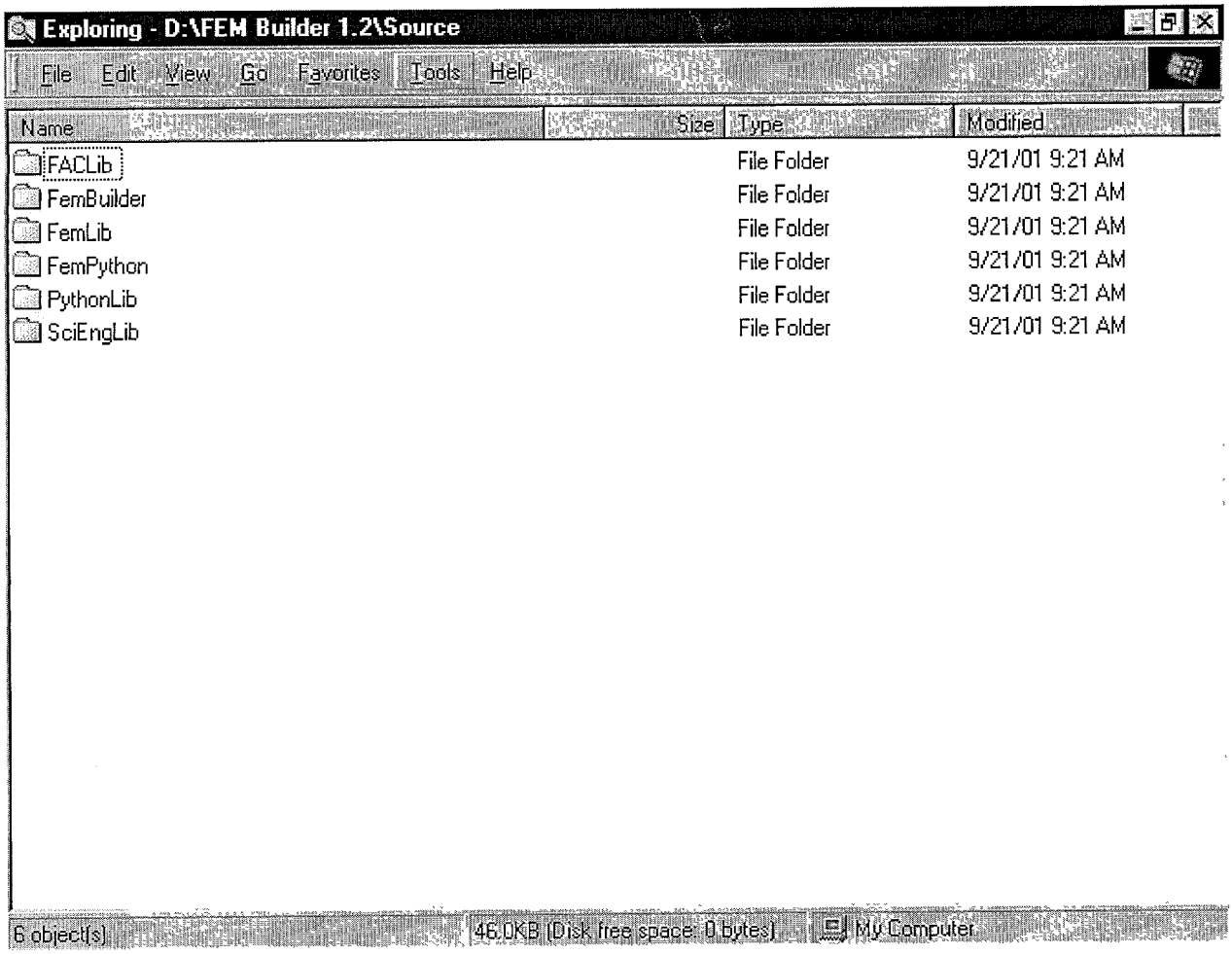

Exploring - D:\FEM Builder 1.2\Doc\Test
File Edit View Go Favorites Tools Help
Name Size Type Modified
FEMBuilderTestStandardEdgeNotched2D doc 164KB Microsoft Word Doc... 9/18/01 7:12 AM
FEMBuilderTestStandardEdgeNotched3D.doc 217KB Microsoft Word Doc... 9/18/01 7:12 AM
FEMBuilderTestStandardFluidCFD1.doc 36KB Microsoft Word Doc... 9/18/01 7:12 AM
FEMBuilderTestStandardHeat0.doc 44KB Microsoft Word Doc... 9/18/01 7:12 AM
FEMBuilderTestStandardHeat1.doc 47KB Microsoft Word Doc... 9/18/01 7:12 AM
FEMBuilderTestStandardHTAxisym1.doc 50KB Microsoft Word Doc... 9/18/01 7:12 AM
FEMBuilderTestStandardNozzle1.doc 38KB Microsoft Word Doc... 9/18/01 7:12 AM
FEMBuilderTestStandardPlaneE1.doc 25KB Microsoft Word Doc... 9/18/01 7:12 AM
FEMBuilderTestStandardPlaneE2.doc 30KB Microsoft Word Doc... 9/18/01 7:12 AM
FEMBuilderTestStandardPlaneS1.doc 25KB Microsoft Word Doc... 9/18/01 7:12 AM
FEMBuilderTestStandardPlaneS2.doc 30KB Microsoft Word Doc... 9/18/01 7:12 AM
FEMBuilderTestStandardSolid1L.doc 26KB Microsoft Word Doc... 9/18/01 7:12 AM
FEMBuilderTestStandardSolid1Q.doc 27KB Microsoft Word Doc... 9/18/01 7:12 AM
FEMBuilderTestStandardSolidCFD1.doc 36KB Microsoft Word Doc... 9/18/01 7:12 AM
FEMBuilderTestSubModel_1.doc 53KB Microsoft Word Doc... 9/18/01 12:26 PM
FEMBuilderTestSubModel_2.doc 51KB Microsoft Word Doc... 9/18/01 7:12 AM
FEMBuilderTestSubModel_3.doc 54KB Microsoft Word Doc... 9/18/01 7:12 AM
FEMBuilderTestThermalAblation_1.doc 68KB Microsoft Word Doc... 9/18/01 12:26 PM
FEMBuilderTestThermalAblation_2.doc 100KB Microsoft Word Doc... 9/18/01 7:12 AM
FEMBuilderTestToolsOptions_1.doc 28KB Microsoft Word Doc... 9/18/01 7:12 AM
FEMBuilderTestViewBlankUnblank_1 doc 65KB Microsoft Word Doc... 9/18/01 7:12 AM
FEMBuilderTestViewDisplaySettings_1.doc 252KB Microsoft Word Doc... 9/18/01 7:12 AM
FEMBuilderTestViewOrientation_1.doc 41KB Microsoft Word Doc... 9/18/01 7:12 AM
210 object(s) 13.6MB (Disk free space: 0 bytes) My Computer

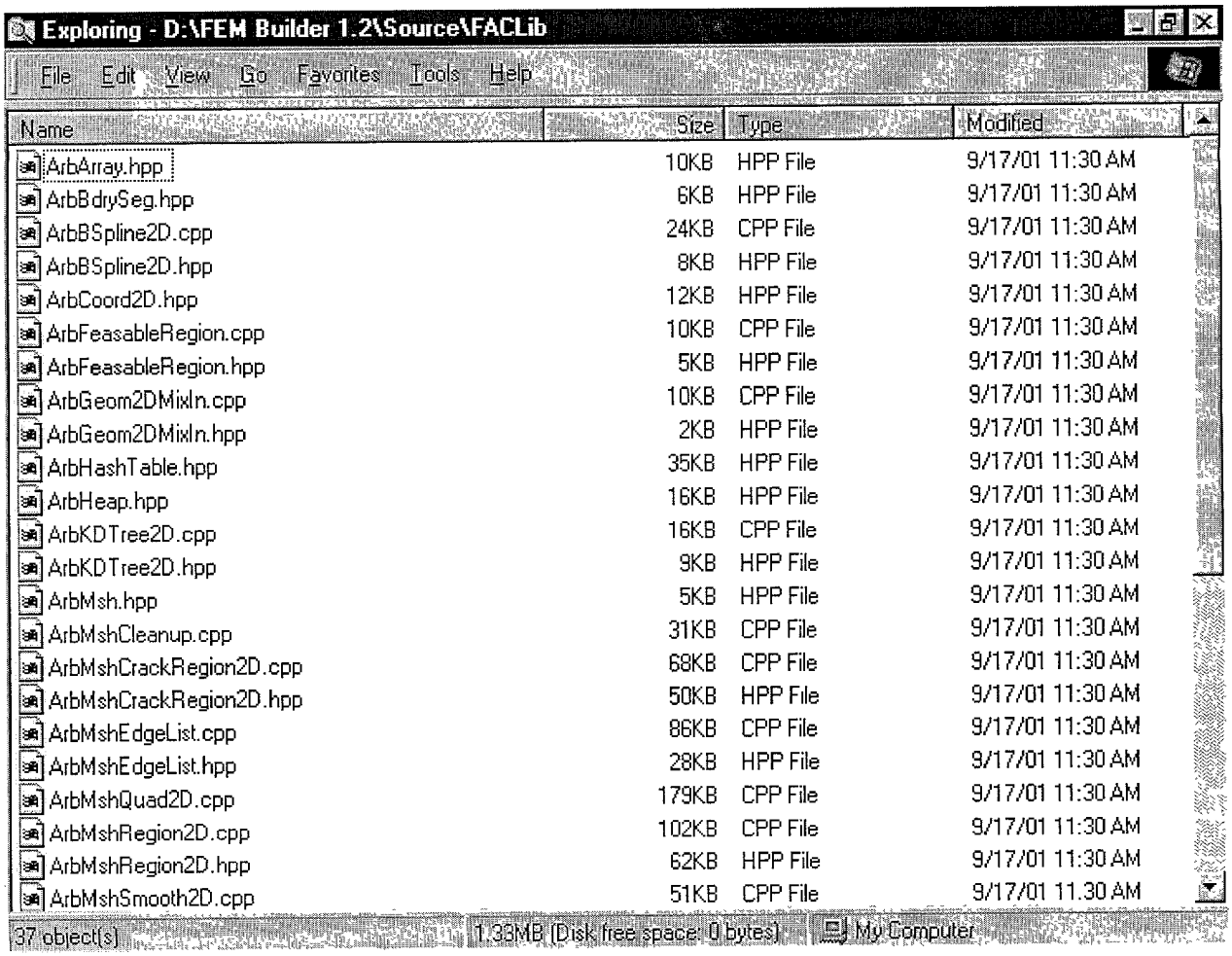

Exploring - D:\FEM Builder 1.2\Doc\Test
File Edit View Go Favorites Tools Help
Name Size Type Modified
FEMBuilderTestStandardHeat1.doc 47KB Microsoft Word Doc... 9/18/01 7:12 AM
FEMBuilderTestStandardHTAxisym1.doc 50KB Microsoft Word Doc... 9/18/01 7:12 AM
FEMBuilderTestStandardNozzle1.doc 38KB Microsoft Word Doc... 9/18/01 7:12 AM
FEMBuilderTestStandardPlaneE1.doc 25KB Microsoft Word Doc... 9/18/01 7:12 AM
FEMBuilderTestStandardPlaneE2.doc 30KB Microsoft Word Doc... 9/18/01 7:12 AM
FEMBuilderTestStandardPlaneS1.doc 25KB Microsoft Word Doc... 9/18/01 7:12 AM
FEMBuilderTestStandardPlaneS2.doc 30KB Microsoft Word Doc... 9/18/01 7:12 AM
FEMBuilderTestStandardSolid1L.doc 26KB Microsoft Word Doc... 9/18/01 7:12 AM
FEMBuilderTestStandardSolid1Q.doc 27KB Microsoft Word Doc... 9/18/01 7:12 AM
FEMBuilderTestStandardSolidCFD1.doc 36KB Microsoft Word Doc... 9/18/01 7:12 AM
FEMBuilderTestSubModel_1.doc 53KB Microsoft Word Doc... 9/18/01 12:26 PM
FEMBuilderTestSubModel_2.doc 51KB Microsoft Word Doc... 9/18/01 7:12 AM
FEMBuilderTestSubModel_3.doc 54KB Microsoft Word Doc... 9/18/01 7:12 AM
FEMBuilderTestThermalAblation_1.doc 68KB Microsoft Word Doc... 9/18/01 12:26 PM
FEMBuilderTestThermalAblation_2.doc 100KB Microsoft Word Doc... 9/18/01 7:12 AM
FEMBuilderTestToolsOptions_1.doc 28KB Microsoft Word Doc... 9/18/01 7:12 AM
FEMBuilderTestViewBlankUnblank_1.doc 65KB Microsoft Word Doc... 9/18/01 7:12 AM
FEMBuilderTestViewDisplaySettings_1.doc 252KB Microsoft Word Doc... 9/18/01 7:12 AM
FEMBuilderTestViewOrientation_1.doc 41KB Microsoft Word Doc... 9/18/01 7:12 AM
FEMBuilderTestViewOrientation_2.doc 228KB Microsoft Word Doc... 9/18/01 7:12 AM
FEMBuilderTestViewOrientation_3.doc 69KB Microsoft Word Doc... 9/18/01 7:12 AM
FEMBuilderTestViewScreenOptions_1.doc 59KB Microsoft Word Doc... 9/18/01 7:12 AM
210 object(s)
13.6MB (Disk free space: 0 bytes)
My Computer

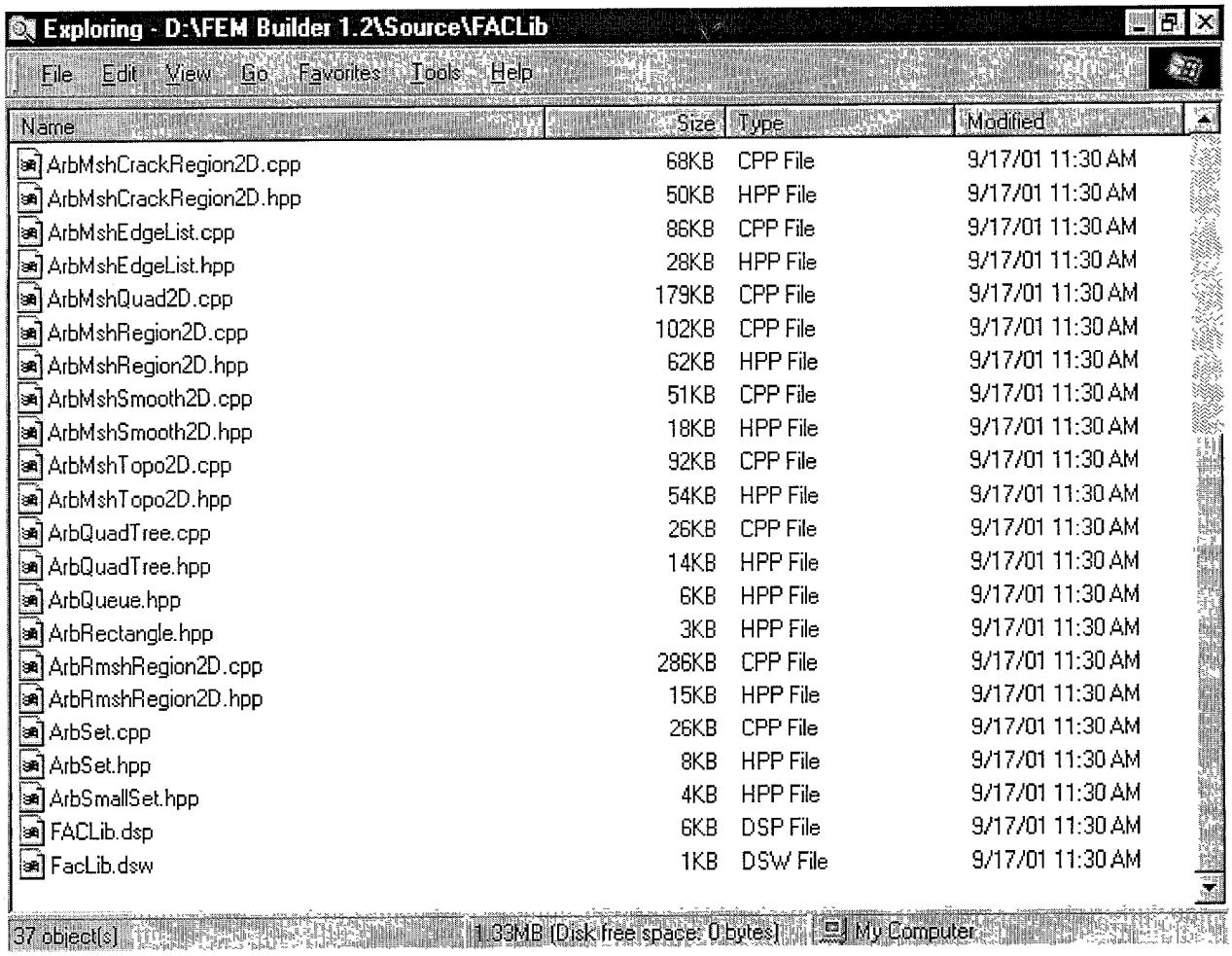

Exploring - D:\FEM Builder 1.2\Doc\User
File Edit View Go Favorites Tools Help
Name Size Type Modified
FEMBuilderPython.doc 410KB Microsoft Word Doc. . 9/17/01 11:34 AM
FemBuilderSolvers.doc 23KB Microsoft Word Doc... 9/17/01 11:34 AM
FemBuilderSolversLess2D.doc 400KB Microsoft Word Doc... 9/17/01 11:34 AM
FEMBuilderTraining.doc 597KB Microsoft Word Doc... 9/19/01 11:54 AM
FEMBuilderUser.doc 111KB Microsoft Word Doc... 9/17/01 11:34 AM
FEMBuilderUserAbaqusInterface.doc 54KB Microsoft Word Doc... 9/17/01 11:34 AM
FEMBuilderUserAnsysInterface.doc 51KB Microsoft Word Doc... 9/17/01 11:34 AM
FEMBuilderUserAppendixAbaqusSupport.doc 111KB Microsoft Word Doc... 9/17/01 11:34 AM
FEMBuilderUserAppendixAnsysSupport.doc 173KB Microsoft Word Doc... 9/17/01 11:34 AM
FEMBuilderUserAppendixElementLibrary.doc 256KB Microsoft Word Doc... 9/17/01 11:34 AM
FEMBuilderUserAppendixMatProperties.doc 40KB Microsoft Word Doc... 9/17/01 11:34 AM
FEMBuilderUserAppendixSelection.doc 65KB Microsoft Word Doc... 9/17/01 11:34 AM
FEMBuilderUserAscharInterface.doc 24KB Microsoft Word Doc... 9/17/01 11:34 AM
FEMBuilderUserCDCA-CCMInterface.doc 39KB Microsoft Word Doc... 9/17/01 11:34 AM
FEMBuilderUserChemicalProperties.doc 25KB Microsoft Word Doc... 9/17/01 11:34 AM
FEMBuilderUserCreateFEM.doc 102KB Microsoft Word Doc... 9/17/01 11:34 AM
FEMBuilderUserCreateGeometry.doc 38KB Microsoft Word Doc... 9/17/01 11:34 AM
FEMBuilderUserDeleteFEM.doc 25KB Microsoft Word Doc... 9/17/01 11:34 AM
FEMBuilderUserDeleteGeometry.doc 25KB Microsoft Word Doc... 9/17/01 11:34 AM
FEMBuilderUserEditFEM.doc 46KB Microsoft Word Doc... 9/17/01 11:34 AM
FEMBuilderUserEditGeometry.doc 32KB Microsoft Word Doc... 9/17/01 11:34 AM
FEMBuilderUserExecution.doc 48KB Microsoft Word Doc... 9/17/01 11:34 AM
FEMBuilderUserFluentInterface.doc 43KB Microsoft Word Doc... 9/17/01 11:34 AM
44 object(s) 3.98MB (Disk free space: 0 bytes) My Computer

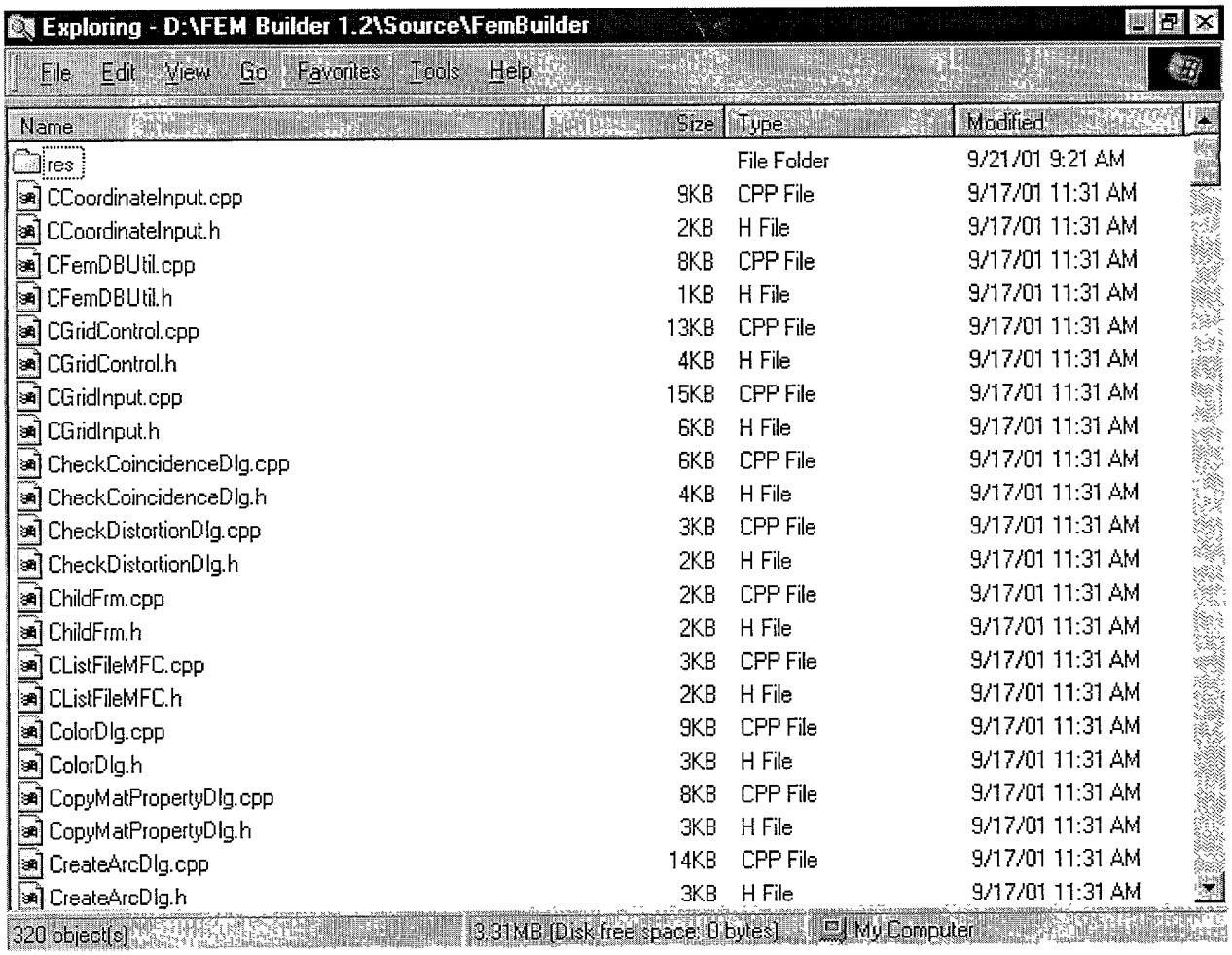

Exploring - D:\FEM Builder 1.2\Doc\User
File Edit View Go Favorites Tools Help
Name Size Type Modified
FEMBuilderUserFluentInterface.doc 43KB Microsoft Word Doc... 9/17/01 11:34 AM
FEMBuilderUserGridGenInterface.doc 23KB Microsoft Word Doc... 9/17/01 11:34 AM
FEMBuilderUserIdeasMSInterface.doc 30KB Microsoft Word Doc... 9/17/01 11:34 AM
FEMBuilderUserIntroduction.doc 43KB Microsoft Word Doc... 9/17/01 11:34 AM
FEMBuilderUserListFEM.doc 25KB Microsoft Word Doc... 9/17/01 11:34 AM
FEMBuilderUserListGeometry doc 26KB Microsoft Word Doc... 9/17/01 11:34 AM
FEMBuilderUserMaterialProperties.doc 29KB Microsoft Word Doc... 9/17/01 11:34 AM
FEMBuilderUserMeshGeneration.doc 148KB Microsoft Word Doc... 9/17/01 11:34 AM
FEMBuilderUserOverview.doc 41KB Microsoft Word Doc... 9/17/01 11:34 AM
FEMBuilderUserPatranInterface.doc 64KB Microsoft Word Doc... 9/17/01 11:34 AM
FEMBuilderUserPostFractureMechanics.doc 181KB Microsoft Word Doc... 9/17/01 11:34 AM
FEMBuilderUserPostResultCalculation.doc 28KB Microsoft Word Doc... 9/17/01 11:34 AM
FEMBuilderUserPostResultDisplay.doc 40KB Microsoft Word Doc... 9/17/01 11:34 AM
FEMBuilderUserPostXYPlotParameters.doc 27KB Microsoft Word Doc... 9/17/01 11:34 AM
FEMBuilderUserReadSharpInterface.doc 26KB Microsoft Word Doc... 9/17/01 11:34 AM
FEMBuilderUserRecessInterface.doc 45KB Microsoft Word Doc... 9/17/01 11:34 AM
FEMBuilderUserSindaInterface.doc 42KB Microsoft Word Doc... 9/17/01 11:34 AM
FEMBuilderUserSinGenInterface.doc 23KB Microsoft Word Doc... 9/17/01 11:34 AM
FEMBuilderUserTexChemInterface.doc 24KB Microsoft Word Doc... 9/17/01 11:34 AM
FEMBuilderUserThermalAblation.doc 306KB Microsoft Word Doc... 9/17/01 11:34 AM
FEMBuilderUserTools.doc 151KB Microsoft Word Doc... 9/17/01 11:34 AM
FEMBuilderUserView.doc 30KB Microsoft Word Doc... 9/17/01 11:34 AM
44 object(s) 3.98MB (Disk free space: 0 bytes) My Computer

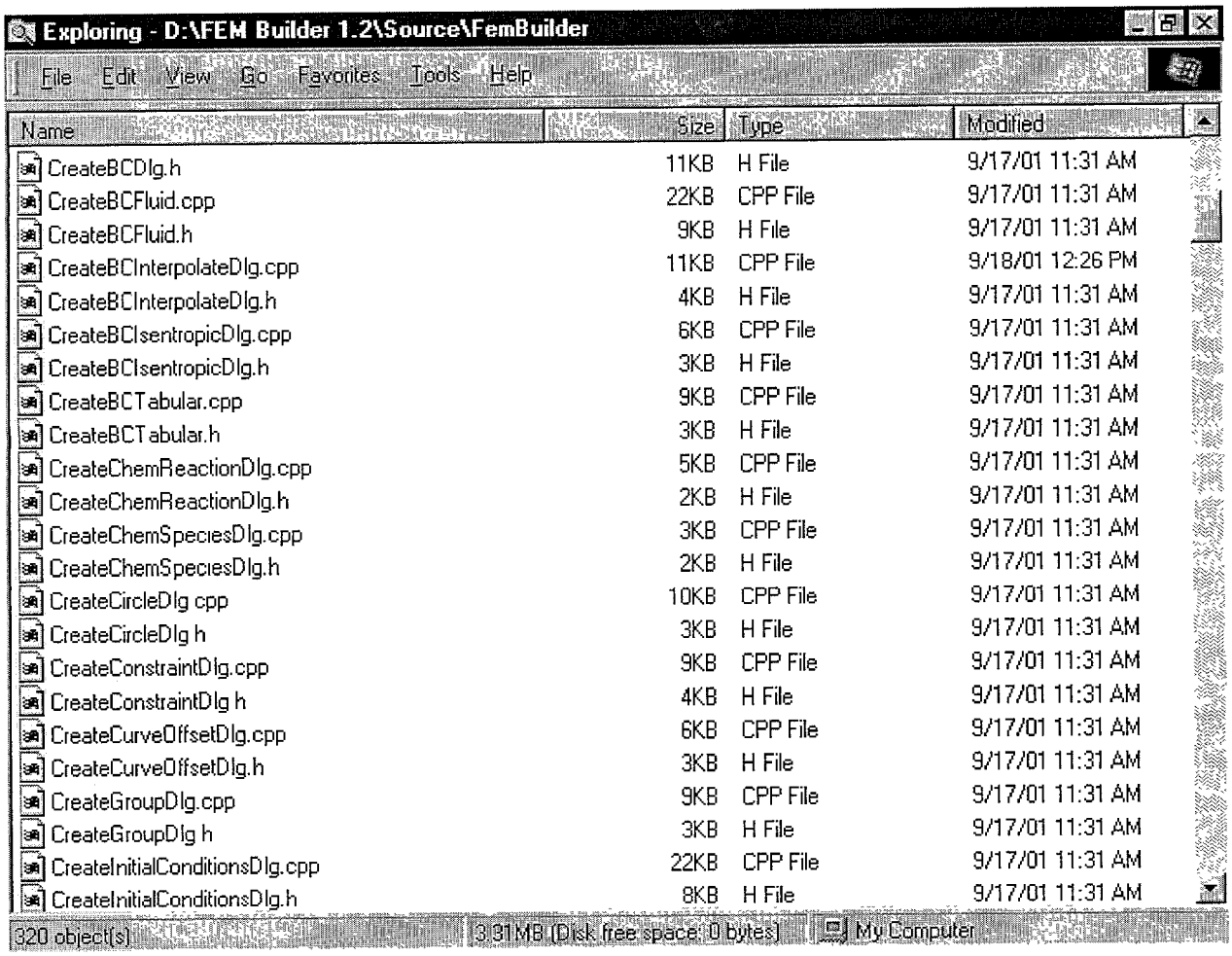

Exploring - D:\FEM Builder 1.2\Python
File Edit View Go Favorites Tools Help
Name Size Type Modified
CExecute.py 9KB Python File 9/17/01 11:34 AM
CExecuteAbaqus.py 8KB Python File 9/17/01 11:34 AM
CExecuteFluent.py 13KB Python File 9/17/01 11:34 AM
CSolve.py 26KB Python File 9/17/01 11:34 AM
CSolveCoupled.py 20KB Python File 9/17/01 11:34 AM
CSolveFSI.py 2KB Python File 9/17/01 11:34 AM
CSolveFSILess2D.py 9KB Python File 9/17/01 11:34 AM
FemHostTable Example.txt 2KB Text Document 9/19/01 8:36 AM
FemHostTable.txt 3KB Text Document 9/19/01 8:35 AM
FemPythonExec.py 28KB Python File 9/17/01 11:34 AM
FemPythonFileIO.py 12KB Python File 9/17/01 11:34 AM
FemPythonTools.py 13KB Python File 9/17/01 11:34 AM
12 object(s) 137KB (Disk free space: 0 bytes) My Computer

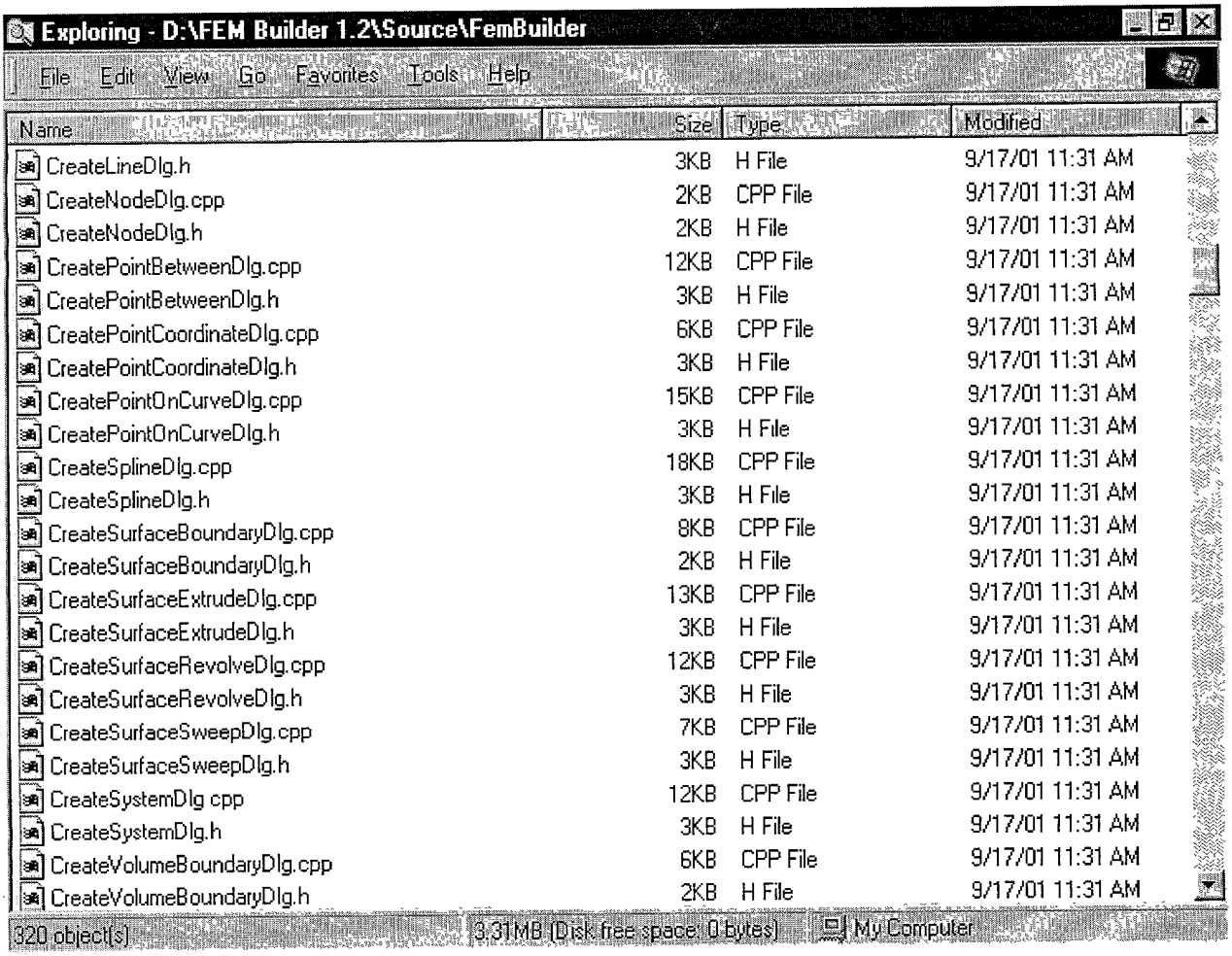

Exploring - D:\FEM Builder 1.2\Source
File Edit View Go Favorites Tools Help
Name Size Type Modified
FACLib File Folder 9/21/01 9:21 AM
FemBuilder File Folder 9/21/01 9:21 AM
FemLib File Folder 9/21/01 9:21 AM
FemPython File Folder 9/21/01 9:21 AM
PythonLib File Folder 9/21/01 9:21 AM
SciEngLib File Folder 9/21/01 9:21 AM
6 object(s)
46.0KB (Disk free space: 0 bytes)
My Computer

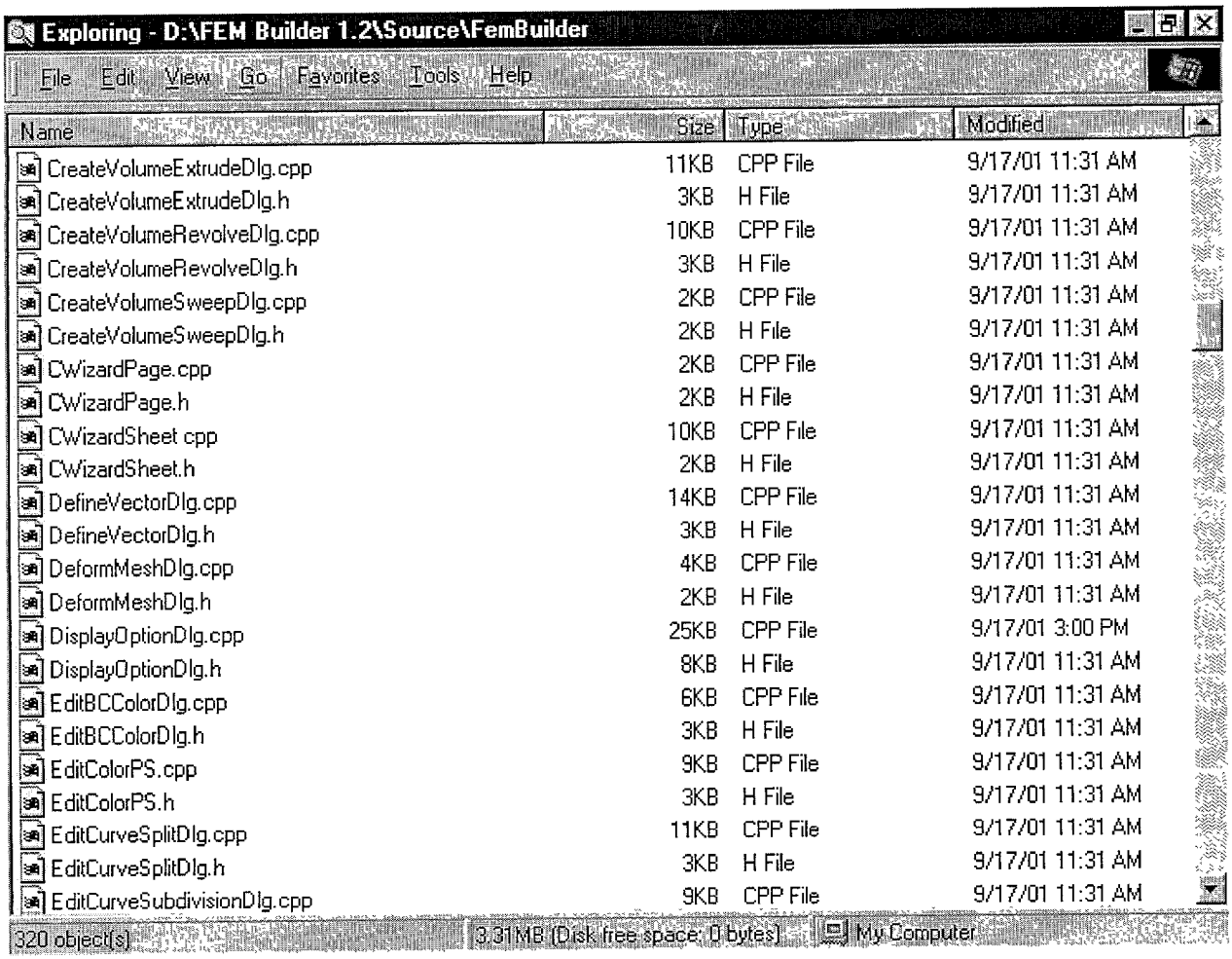

Exploring - D:\FEM Builder 1.2\Source\FACLib
File Edit View Go Favorites Tools Help
Name Size Type Modified
ArbArray.hpp 10KB HPP File 9/17/01 11:30 AM
ArbBdrySeg.hpp 6KB HPP File 9/17/01 11:30 AM
ArbBSpline2D.cpp 24KB CPP File 9/17/01 11:30 AM
ArbBSpline2D.hpp 8KB HPP File 9/17/01 11:30 AM
ArbCoord2D.hpp 12KB HPP File 9/17/01 11:30 AM
ArbFeasableRegion.cpp 10KB CPP File 9/17/01 11:30 AM
ArbFeasableRegion.hpp 5KB HPP File 9/17/01 11:30 AM
ArbGeom2DMixIn.cpp 10KB CPP File 9/17/01 11:30 AM
ArbGeom2DMixIn.hpp 2KB HPP File 9/17/01 11:30 AM
ArbHashTable.hpp 35KB HPP File 9/17/01 11:30 AM
ArbHeap.hpp 16KB HPP File 9/17/01 11:30 AM
ArbKDTree2D.cpp 16KB CPP File 9/17/01 11:30 AM
ArbKDTree2D.hpp 9KB HPP File 9/17/01 11:30 AM
ArbMsh.hpp 5KB HPP File 9/17/01 11:30 AM
ArbMshCleanup.cpp 31KB CPP File 9/17/01 11:30 AM
ArbMshCrackRegion2D.cpp 68KB CPP File 9/17/01 11:30 AM
ArbMshCrackRegion2D.hpp 50KB HPP File 9/17/01 11:30 AM
ArbMshEdgeList.cpp 86KB CPP File 9/17/01 11:30 AM
ArbMshEdgeList.hpp 28KB HPP File 9/17/01 11:30 AM
ArbMshQuad2D.cpp 179KB CPP File 9/17/01 11:30 AM
ArbMshRegion2D.cpp 102KB CPP File 9/17/01 11:30 AM
ArbMshRegion2D.hpp 62KB HPP File 9/17/01 11:30 AM
ArbMshSmooth2D.cpp 51KB CPP File 9/17/01 11:30 AM
37 object(s) 1.38MB (Disk free space: 0 bytes) My Computer

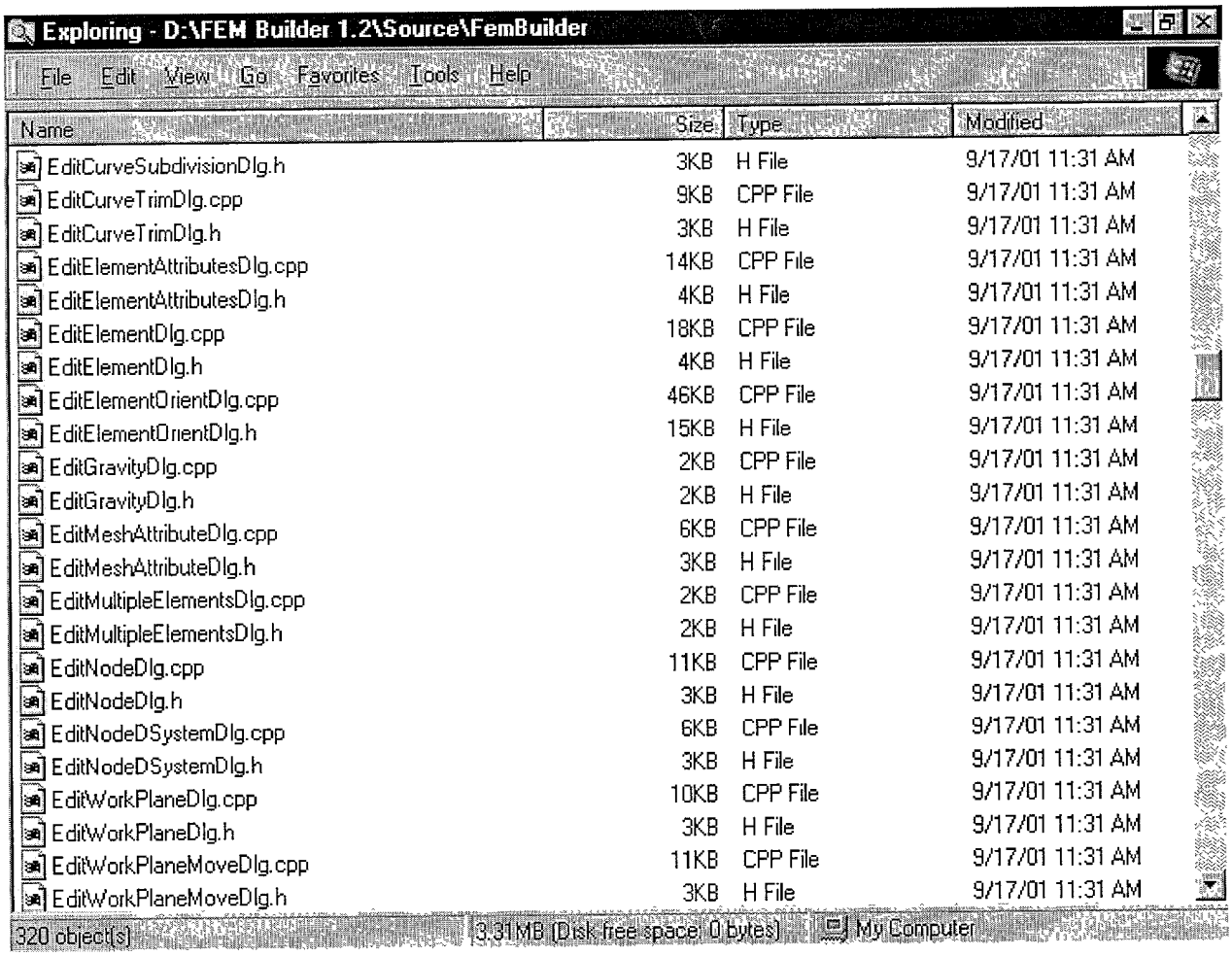

Exploring - D:\FEM Builder 1.2\Source\FACLib
File Edit View Go Favorites Tools Help
Name Size Type Modified
ArbMshCrackRegion2D.cpp 68KB CPP File 9/17/01 11:30 AM
ArbMshCrackRegion2D.hpp 50KB HPP File 9/17/01 11:30 AM
ArbMshEdgeList.cpp 86KB CPP File 9/17/01 11:30 AM
ArbMshEdgeList.hpp 28KB HPP File 9/17/01 11:30 AM
ArbMshQuad2D.cpp 179KB CPP File 9/17/01 11:30 AM
ArbMshRegion2D.cpp 102KB CPP File 9/17/01 11:30 AM
ArbMshRegion2D.hpp 62KB HPP File 9/17/01 11:30 AM
ArbMshSmooth2D.cpp 51KB CPP File 9/17/01 11:30 AM
ArbMshSmooth2D.hpp 18KB HPP File 9/17/01 11:30 AM
ArbMshTopo2D.cpp 92KB CPP File 9/17/01 11:30 AM
ArbMshTopo2D.hpp 54KB HPP File 9/17/01 11:30 AM
ArbQuadTree.cpp 26KB CPP File 9/17/01 11:30 AM
ArbQuadTree.hpp 14KB HPP File 9/17/01 11:30 AM
ArbQueue.hpp 6KB HPP File 9/17/01 11:30 AM
ArbRectangle.hpp 3KB HPP File 9/17/01 11:30 AM
ArbRmshRegion2D.cpp 286KB CPP File 9/17/01 11:30 AM
ArbRmshRegion2D.hpp 15KB HPP File 9/17/01 11:30 AM
ArbSet.cpp 26KB CPP File 9/17/01 11:30 AM
ArbSet.hpp 8KB HPP File 9/17/01 11:30 AM
ArbSmallSet.hpp 4KB HPP File 9/17/01 11:30 AM
FACLib.dsp 6KB DSP File 9/17/01 11:30 AM
FacLib.dsw 1KB DSW File 9/17/01 11:30 AM
37 object(s) 1.33MB (Disk free space: 0 bytes) My Computer

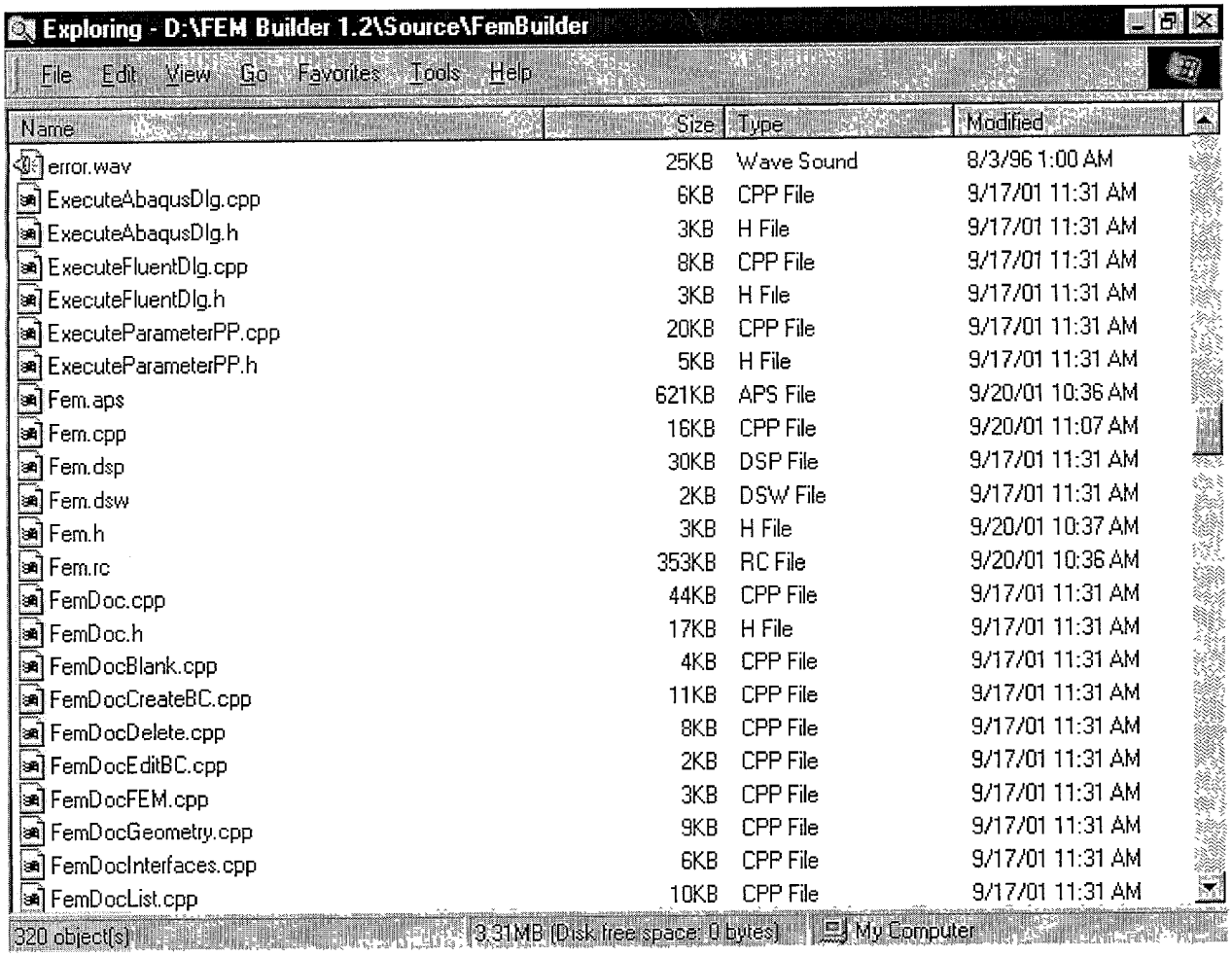

Exploring - D:\FEM Builder 1.2\Source\FemBuilder
File Edit View Go Favorites Tools Help
Name Size Type Modified
res File Folder 9/21/01 9:21 AM
CCoordinateInput.cpp 9KB CPP File 9/17/01 11:31 AM
CCoordinateInput.h 2KB H File 9/17/01 11:31 AM
CFemDBUtil.cpp 8KB CPP File 9/17/01 11:31 AM
CFemDBUtil.h 1KB H File 9/17/01 11:31 AM
CGridControl.cpp 13KB CPP File 9/17/01 11:31 AM
CGridControl.h 4KB H File 9/17/01 11:31 AM
CGridInput.cpp 15KB CPP File 9/17/01 11:31 AM
CGridInput.h 6KB H File 9/17/01 11:31 AM
CheckCoincidenceDlg.cpp 6KB CPP File 9/17/01 11:31 AM
CheckCoincidenceDlg.h 4KB H File 9/17/01 11:31 AM
CheckDistortionDlg.cpp 3KB CPP File 9/17/01 11:31 AM
CheckDistortionDlg.h 2KB H File 9/17/01 11:31 AM
ChildFrm.cpp 2KB CPP File 9/17/01 11:31 AM
ChildFrm.h 2KB H File 9/17/01 11:31 AM
CListFileMFC.cpp 3KB CPP File 9/17/01 11:31 AM
CListFileMFC.h 2KB H File 9/17/01 11:31 AM
ColorDlg.cpp 9KB CPP File 9/17/01 11:31 AM
ColorDlg.h 3KB H File 9/17/01 11:31 AM
CopyMatPropertyDlg.cpp 8KB CPP File 9/17/01 11:31 AM
CopyMatPropertyDlg.h 3KB H File 9/17/01 11:31 AM
CreateArcDlg.cpp 14KB CPP File 9/17/01 11:31 AM
CreateArcDlg.h 3KB H File 9/17/01 11:31 AM
320 object(s) 3.31 MB (Disk free space: 0 bytes) My Computer

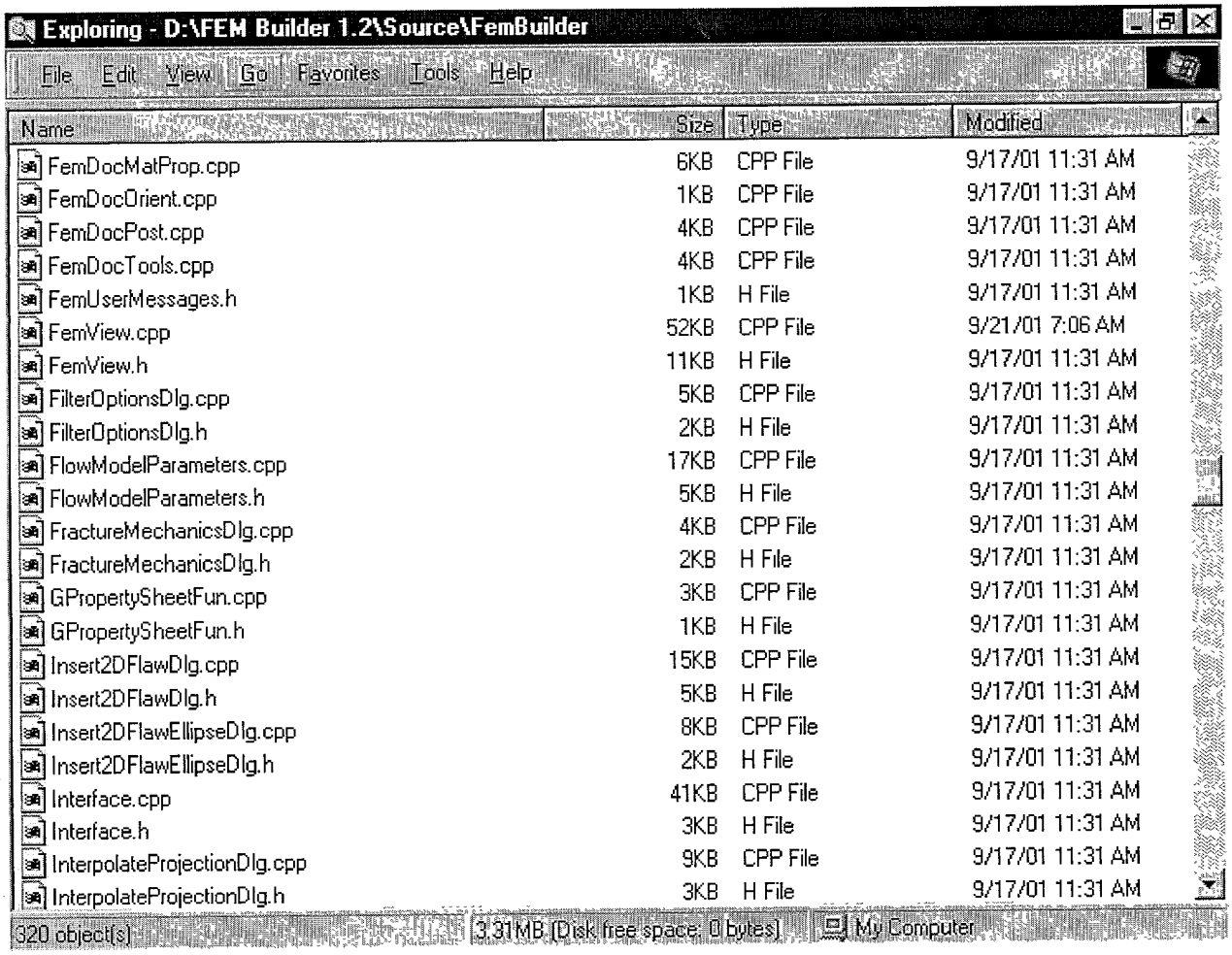

Exploring - D:\FEM Builder 1.2\Source\FemBuilder
File Edit View Go Favorites Tools Help
Name Size Type Modified
CreateBCDlg.h 11KB H File 9/17/01 11:31 AM
CreateBCFluid.cpp 22KB CPP File 9/17/01 11:31 AM
CreateBCFluid.h 9KB H File 9/17/01 11:31 AM
CreateBCInterpolateDlg.cpp 11KB CPP File 9/18/01 12:26 PM
CreateBCInterpolateDlg.h 4KB H File 9/17/01 11:31 AM
CreateBCIsentropicDlg.cpp 6KB CPP File 9/17/01 11:31 AM
CreateBCIsentropicDlg.h 3KB H File 9/17/01 11:31 AM
CreateBCTabular.cpp 9KB CPP File 9/17/01 11:31 AM
CreateBCTabular.h 3KB H File 9/17/01 11:31 AM
CreateChemReactionDlg.cpp 5KB CPP File 9/17/01 11:31 AM
CreateChemReactionDlg.h 2KB H File 9/17/01 11:31 AM
CreateChemSpeciesDlg.cpp 3KB CPP File 9/17/01 11:31 AM
CreateChemSpeciesDlg.h 2KB H File 9/17/01 11:31 AM
CreateCircleDlg cpp 10KB CPP File 9/17/01 11:31 AM
CreateCircleDlg h 3KB H File 9/17/01 11:31 AM
CreateConstraintDlg.cpp 9KB CPP File 9/17/01 11:31 AM
CreateConstraintDlg h 4KB H File 9/17/01 11:31 AM
CreateCurveOffsetDlg.cpp 6KB CPP File 9/17/01 11:31 AM
CreateCurveOffsetDlg.h 3KB H File 9/17/01 11:31 AM
CreateGroupDlg.cpp 9KB CPP File 9/17/01 11:31 AM
CreateGroupDlg h 3KB H File 9/17/01 11:31 AM
CreateInitialConditionsDlg.cpp 22KB CPP File 9/17/01 11:31 AM
CreateInitialConditionsDlg.h 8KB H File 9/17/01 11:31 AM
320 object(s) 3.31 MB (Disk free space: 0 bytes) My Computer

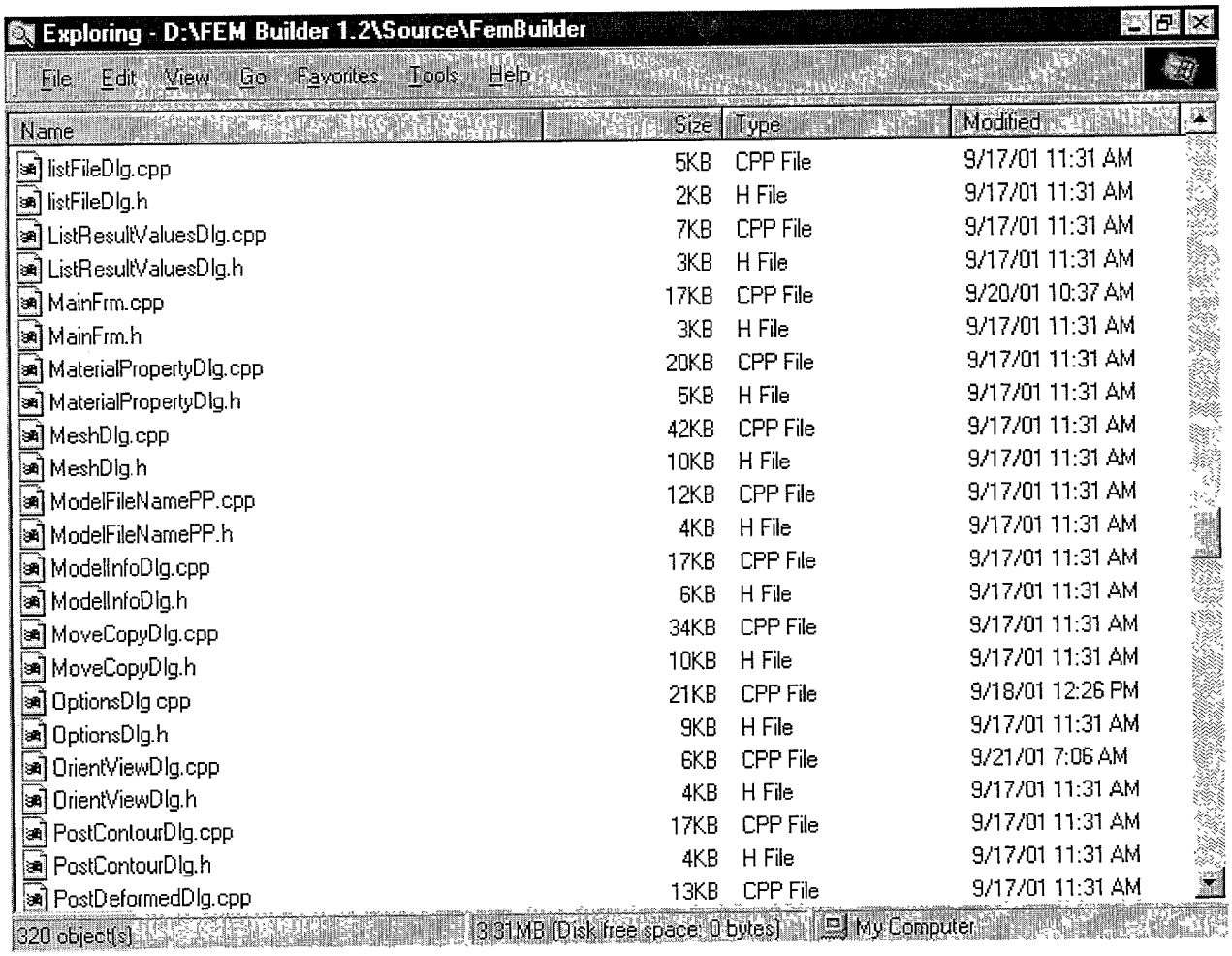

Exploring - D:\FEM Builder 1.2\Source\FemBuilder
File Edit View Go Favorites Tools Help
Name Size Type Modified
CreateLineDlg.h 3KB H File 9/17/01 11:31 AM
CreateNodeDlg.cpp 2KB CPP File 9/17/01 11:31 AM
CreateNodeDlg.h 2KB H File 9/17/01 11:31 AM
CreatePointBetweenDlg.cpp 12KB CPP File 9/17/01 11:31 AM
CreatePointBetweenDlg.h 3KB H File 9/17/01 11:31 AM
CreatePointCoordinateDlg.cpp 6KB CPP File 9/17/01 11:31 AM
CreatePointCoordinateDlg.h 3KB H File 9/17/01 11:31 AM
CreatePointOnCurveDlg.cpp 15KB CPP File 9/17/01 11:31 AM
CreatePointOnCurveDlg.h 3KB H File 9/17/01 11:31 AM
CreateSplineDlg.cpp 18KB CPP File 9/17/01 11:31 AM
CreateSplineDlg.h 3KB H File 9/17/01 11:31 AM
CreateSurfaceBoundaryDlg.cpp 8KB CPP File 9/17/01 11:31 AM
CreateSurfaceBoundaryDlg.h 2KB H File 9/17/01 11:31 AM
CreateSurfaceExtrudeDlg.cpp 13KB CPP File 9/17/01 11:31 AM
CreateSurfaceExtrudeDlg.h 3KB H File 9/17/01 11:31 AM
CreateSurfaceRevolveDlg.cpp 12KB CPP File 9/17/01 11:31 AM
CreateSurfaceRevolveDlg.h 3KB H File 9/17/01 11:31 AM
CreateSurfaceSweepDlg.cpp 7KB CPP File 9/17/01 11:31 AM
CreateSurfaceSweepDlg.h 3KB H File 9/17/01 11:31 AM
CreateSystemDlg cpp 12KB CPP File 9/17/01 11:31 AM
CreateSystemDlg.h 3KB H File 9/17/01 11:31 AM
CreateVolumeBoundaryDlg.cpp 6KB CPP File 9/17/01 11:31 AM
CreateVolumeBoundaryDlg.h 2KB H File 9/17/01 11:31 AM
320 object(s) 3.31 MB (Disk free space: 0 bytes) My Computer

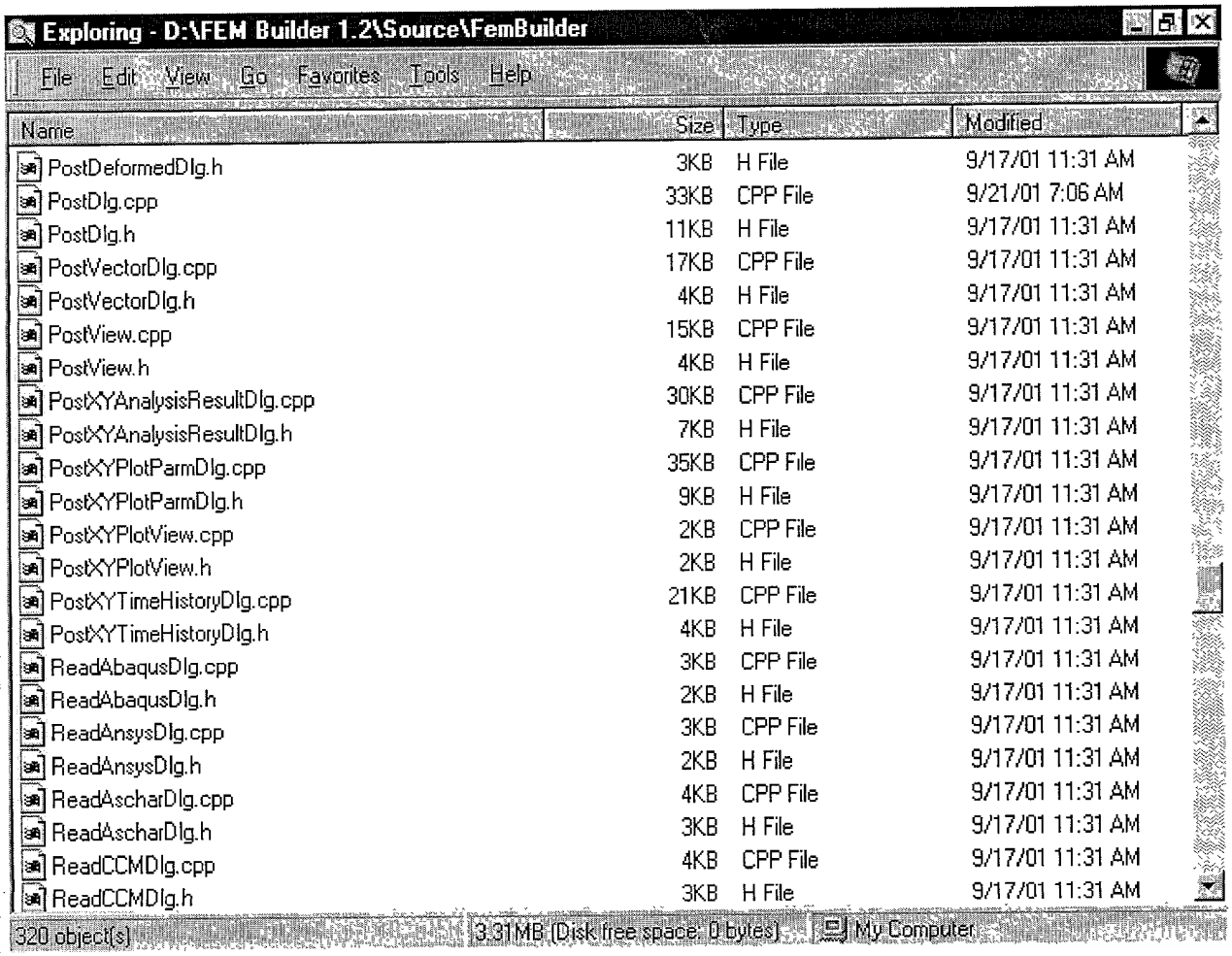

Exploring - D:\FEM Builder 1.2\Source\FemBuilder
File Edit View Go Favorites Tools Help
Name Size Type Modified
CreateVolumeExtrudeDlg.cpp 11KB CPP File 9/17/01 11:31 AM
CreateVolumeExtrudeDlg.h 3KB H File 9/17/01 11:31 AM
CreateVolumeRevolveDlg.cpp 10KB CPP File 9/17/01 11:31 AM
CreateVolumeRevolveDlg.h 3KB H File 9/17/01 11:31 AM
CreateVolumeSweepDlg.cpp 2KB CPP File 9/17/01 11:31 AM
CreateVolumeSweepDlg.h 2KB H File 9/17/01 11:31 AM
CWizardPage.cpp 2KB CPP File 9/17/01 11:31 AM
CWizardPage.h 2KB H File 9/17/01 11:31 AM
CWizardSheet cpp 10KB CPP File 9/17/01 11:31 AM
CWizardSheet.h 2KB H File 9/17/01 11:31 AM
DefineVectorDlg.cpp 14KB CPP File 9/17/01 11:31 AM
DefineVectorDlg.h 3KB H File 9/17/01 11:31 AM
DeformMeshDlg.cpp 4KB CPP File 9/17/01 11:31 AM
DeformMeshDlg.h 2KB H File 9/17/01 11:31 AM
DisplayOptionDlg.cpp 25KB CPP File 9/17/01 3:00 PM
DisplayOptionDlg.h 8KB H File 9/17/01 11:31 AM
EditBCColorDlg.cpp 6KB CPP File 9/17/01 11:31 AM
EditBCColorDlg.h 3KB H File 9/17/01 11:31 AM
EditColorPS.cpp 9KB CPP File 9/17/01 11:31 AM
EditColorPS.h 3KB H File 9/17/01 11:31 AM
EditCurveSplitDlg.cpp 11KB CPP File 9/17/01 11:31 AM
EditCurveSplitDlg.h 3KB H File 9/17/01 11:31 AM
EditCurveSubdivisionDlg.cpp 9KB CPP File 9/17/01 11:31 AM
320 object(s) 3.31MB (Disk free space: 0 bytes) My Computer

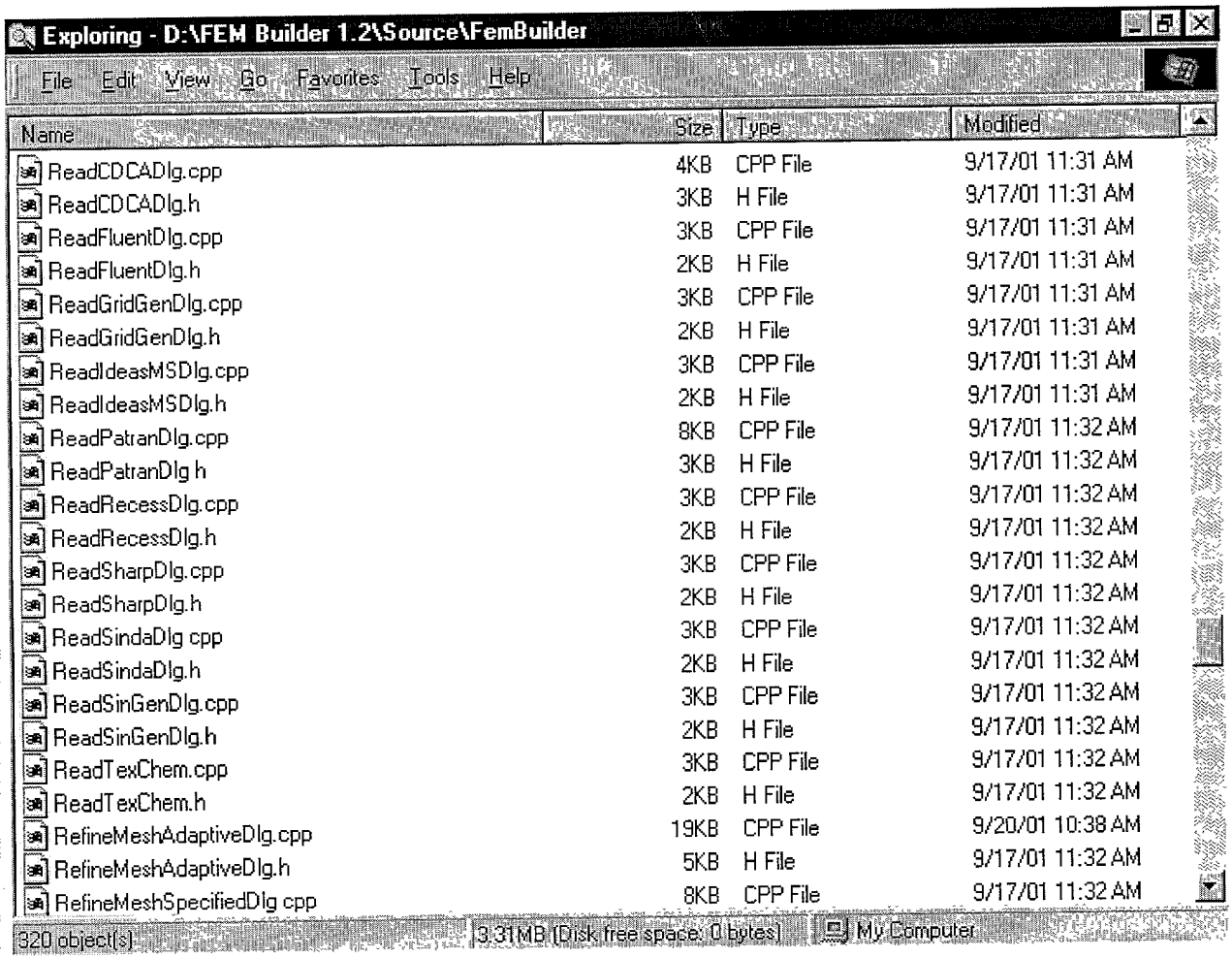

Exploring - D:\FEM Builder 1.2\Source\FemBuilder
File Edit View Go Favorites Tools Help
Name Size Type Modified
EditCurveSubdivisionDlg.h 3KB H File 9/17/01 11:31 AM
EditCurveTrimDlg.cpp 9KB CPP File 9/17/01 11:31 AM
EditCurveTrimDlg.h 3KB H File 9/17/01 11:31 AM
EditElementAttributesDlg.cpp 14KB CPP File 9/17/01 11:31 AM
EditElementAttributesDlg.h 4KB H File 9/17/01 11:31 AM
EditElementDlg.cpp 18KB CPP File 9/17/01 11:31 AM
EditElementDlg.h 4KB H File 9/17/01 11:31 AM
EditElementOrientDlg.cpp 46KB CPP File 9/17/01 11:31 AM
EditElementOrientDlg.h 15KB H File 9/17/01 11:31 AM
EditGravityDlg.cpp 2KB CPP File 9/17/01 11:31 AM
EditGravityDlg.h 2KB H File 9/17/01 11:31 AM
EditMeshAttributeDlg.cpp 6KB CPP File 9/17/01 11:31 AM
EditMeshAttributeDlg.h 3KB H File 9/17/01 11:31 AM
EditMultipleElementsDlg.cpp 2KB CPP File 9/17/01 11:31 AM
EditMultipleElementsDlg.h 2KB H File 9/17/01 11:31 AM
EditNodeDlg.cpp 11KB CPP File 9/17/01 11:31 AM
EditNodeDlg.h 3KB H File 9/17/01 11:31 AM
EditNodeDSystemDlg.cpp 6KB CPP File 9/17/01 11:31 AM
EditNodeDSystemDlg.h 3KB H File 9/17/01 11:31 AM
EditWorkPlaneDlg.cpp 10KB CPP File 9/17/01 11:31 AM
EditWorkPlaneDlg.h 3KB H File 9/17/01 11:31 AM
EditWorkPlaneMoveDlg.cpp 11KB CPP File 9/17/01 11:31 AM
EditWorkPlaneMoveDlg.h 3KB H File 9/17/01 11:31 AM
320 object(s) 3.31MB (Disk free space: 0 bytes) My Computer

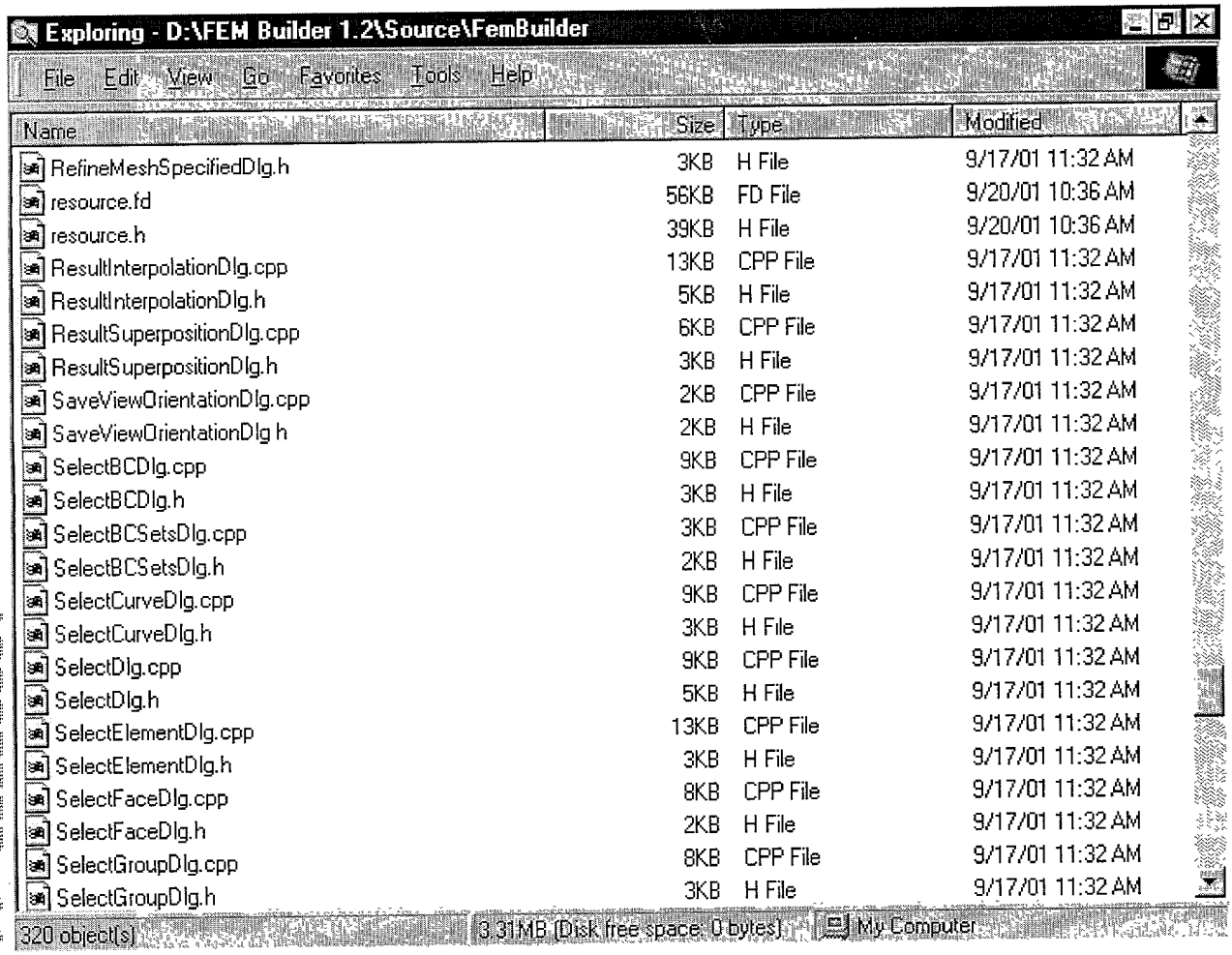

Exploring - D:\FEM Builder 1.2\Source\FemBuilder
File Edit View Go Favorites Tools Help
Name Size Type Modified
error.wav 25KB Wave Sound 8/3/96 1:00 AM
ExecuteAbaqusDlg.cpp 6KB CPP File 9/17/01 11:31 AM
ExecuteAbaqusDlg.h 3KB H File 9/17/01 11:31 AM
ExecuteFluentDlg.cpp 8KB CPP File 9/17/01 11:31 AM
ExecuteFluentDlg.h 3KB H File 9/17/01 11:31 AM
ExecuteParameterPP.cpp 20KB CPP File 9/17/01 11:31 AM
ExecuteParameterPP.h 5KB H File 9/17/01 11:31 AM
Fem.aps 621KB APS File 9/20/01 10:36 AM
Fem.cpp 16KB CPP File 9/20/01 11:07 AM
Fem.dsp 30KB DSP File 9/17/01 11:31 AM
Fem.dsw 2KB DSW File 9/17/01 11:31 AM
Fem.h 3KB H File 9/20/01 10:37 AM
Fem.rc 353KB RC File 9/20/01 10:36 AM
FemDoc.cpp 44KB CPP File 9/17/01 11:31 AM
FemDoc.h 17KB H File 9/17/01 11:31 AM
FemDocBlank.cpp 4KB CPP File 9/17/01 11:31 AM
FemDocCreateBC.cpp 11KB CPP File 9/17/01 11:31 AM
FemDocDelete.cpp 8KB CPP File 9/17/01 11:31 AM
FemDocEditBC.cpp 2KB CPP File 9/17/01 11:31 AM
FemDocFEM.cpp 3KB CPP File 9/17/01 11:31 AM
FemDocGeometry.cpp 9KB CPP File 9/17/01 11:31 AM
FemDocInterfaces.cpp 6KB CPP File 9/17/01 11:31 AM
FemDocList.cpp 10KB CPP File 9/17/01 11:31 AM
320 object(s) 3.31MB (Disk free space: 0 bytes) My Computer

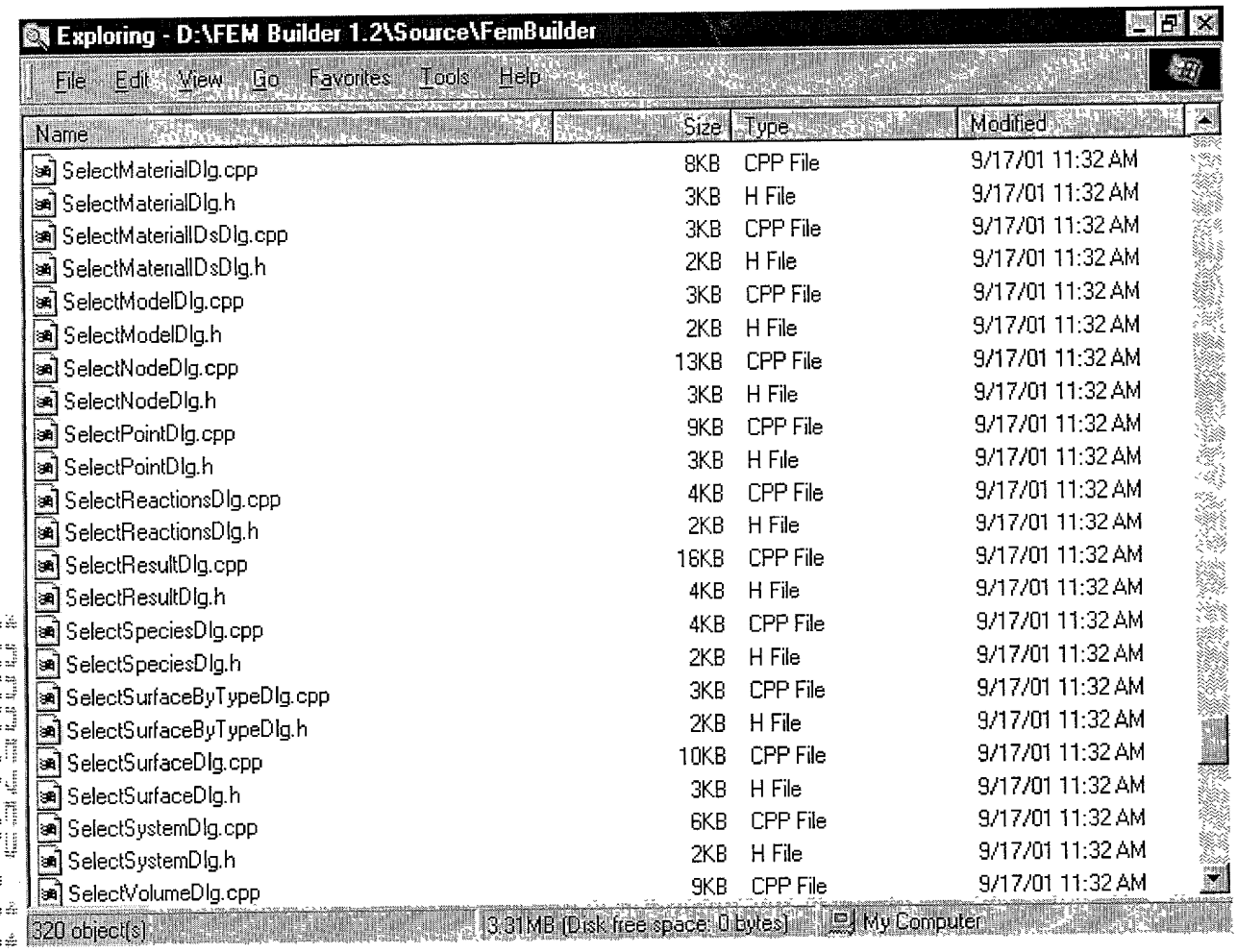

Exploring - D:\FEM Builder 1.2\Source\FemBuilder
File Edit View Go Favorites Tools Help
Name Size Type Modified
FemDocMatProp.cpp 6KB CPP File 9/17/01 11:31 AM
FemDocOrient.cpp 1KB CPP File 9/17/01 11:31 AM
FemDocPost.cpp 4KB CPP File 9/17/01 11:31 AM
FemDocTools.cpp 4KB CPP File 9/17/01 11:31 AM
FemUserMessages.h 1KB H File 9/17/01 11:31 AM
FemView.cpp 52KB CPP File 9/21/01 7:06 AM
FemView.h 11KB H File 9/17/01 11:31 AM
FilterOptionsDlg.cpp 5KB CPP File 9/17/01 11:31 AM
FilterOptionsDlg.h 2KB H File 9/17/01 11:31 AM
FlowModelParameters.cpp 17KB CPP File 9/17/01 11:31 AM
FlowModelParameters.h 5KB H File 9/17/01 11:31 AM
FractureMechanicsDlg.cpp 4KB CPP File 9/17/01 11:31 AM
FractureMechanicsDlg.h 2KB H File 9/17/01 11:31 AM
GPropertySheetFun.cpp 3KB CPP File 9/17/01 11:31 AM
GPropertySheetFun.h 1KB H File 9/17/01 11:31 AM
Insert2DFlawDlg.cpp 15KB CPP File 9/17/01 11:31 AM
Insert2DFlawDlg.h 5KB H File 9/17/01 11:31 AM
Insert2DFlawEllipseDlg.cpp 8KB CPP File 9/17/01 11:31 AM
Insert2DFlawEllipseDlg.h 2KB H File 9/17/01 11:31 AM
Interface.cpp 41KB CPP File 9/17/01 11:31 AM
Interface.h 3KB H File 9/17/01 11:31 AM
InterpolateProjectionDlg.cpp 9KB CPP File 9/17/01 11:31 AM
InterpolateProjectionDlg.h 3KB H File 9/17/01 11:31 AM
320 object(s)
3.31MB (Disk free space: 0 bytes)
My Computer

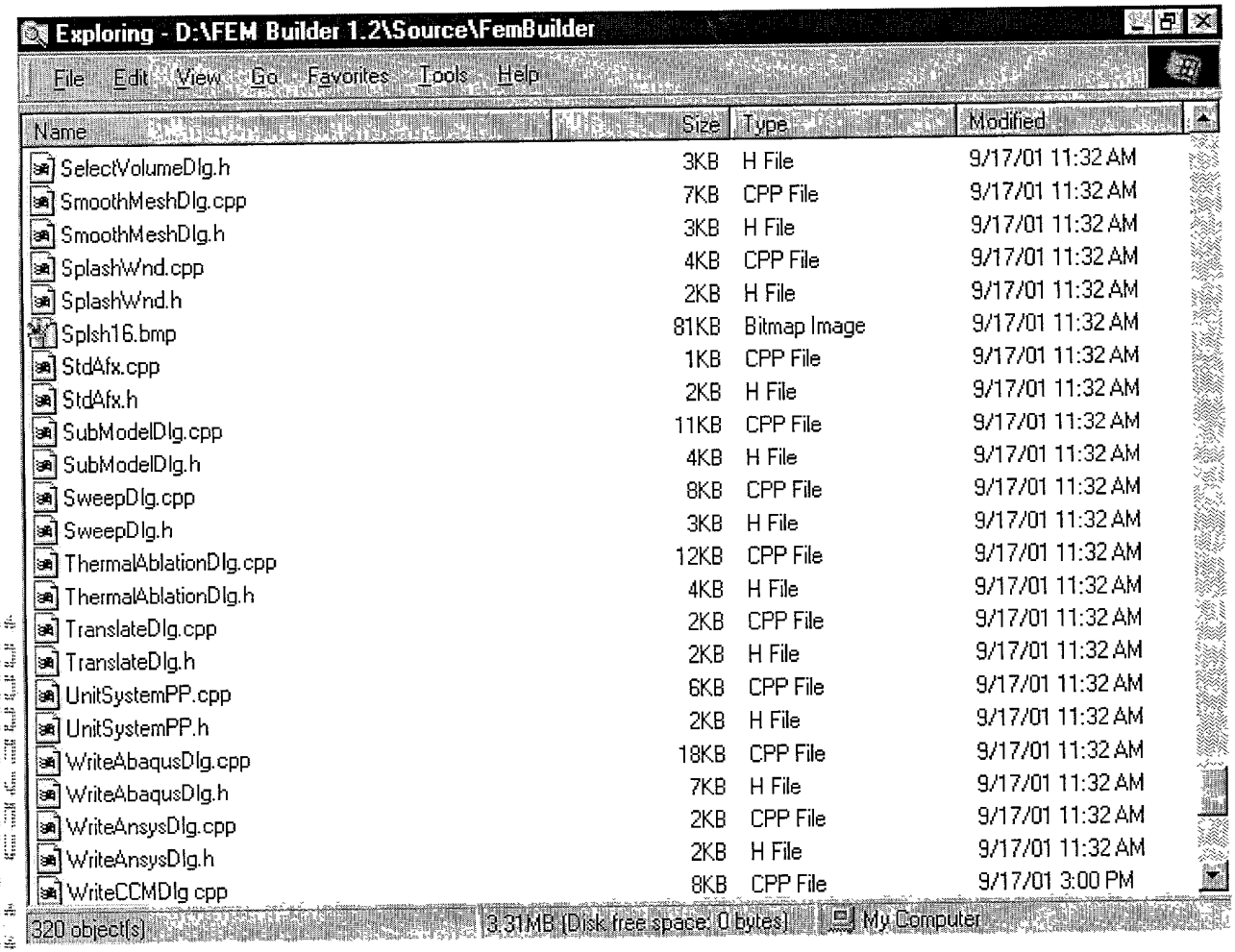

Exploring - D:\FEM Builder 1.2\Source\FemBuilder
File Edit View Go Favorites Tools Help
Name Size Type Modified
listFileDlg.cpp 5KB CPP File 9/17/01 11:31 AM
listFileDlg.h 2KB H File 9/17/01 11:31 AM
ListResultValuesDlg.cpp 7KB CPP File 9/17/01 11:31 AM
ListResultValuesDlg.h 3KB H File 9/17/01 11:31 AM
MainFrm.cpp 17KB CPP File 9/20/01 10:37 AM
MainFrm.h 3KB H File 9/17/01 11:31 AM
MaterialPropertyDlg.cpp 20KB CPP File 9/17/01 11:31 AM
MaterialPropertyDlg.h 5KB H File 9/17/01 11:31 AM
MeshDlg.cpp 42KB CPP File 9/17/01 11:31 AM
MeshDlg.h 10KB H File 9/17/01 11:31 AM
ModelFileNamePP.cpp 12KB CPP File 9/17/01 11:31 AM
ModelFileNamePP.h 4KB H File 9/17/01 11:31 AM
ModelInfoDlg.cpp 17KB CPP File 9/17/01 11:31 AM
ModelInfoDlg.h 6KB H File 9/17/01 11:31 AM
MoveCopyDlg.cpp 34KB CPP File 9/17/01 11:31 AM
MoveCopyDlg.h 10KB H File 9/17/01 11:31 AM
OptionsDlg.cpp 21KB CPP File 9/18/01 12:26 PM
OptionsDlg.h 9KB H File 9/17/01 11:31 AM
OrientViewDlg.cpp 6KB CPP File 9/21/01 7:06 AM
OrientViewDlg.h 4KB H File 9/17/01 11:31 AM
PostContourDlg.cpp 17KB CPP File 9/17/01 11:31 AM
PostContourDlg.h 4KB H File 9/17/01 11:31 AM
PostDeformedDlg.cpp 13KB CPP File 9/17/01 11:31 AM
320 object(s) 8.31MB (Disk free space: 0 bytes) My Computer

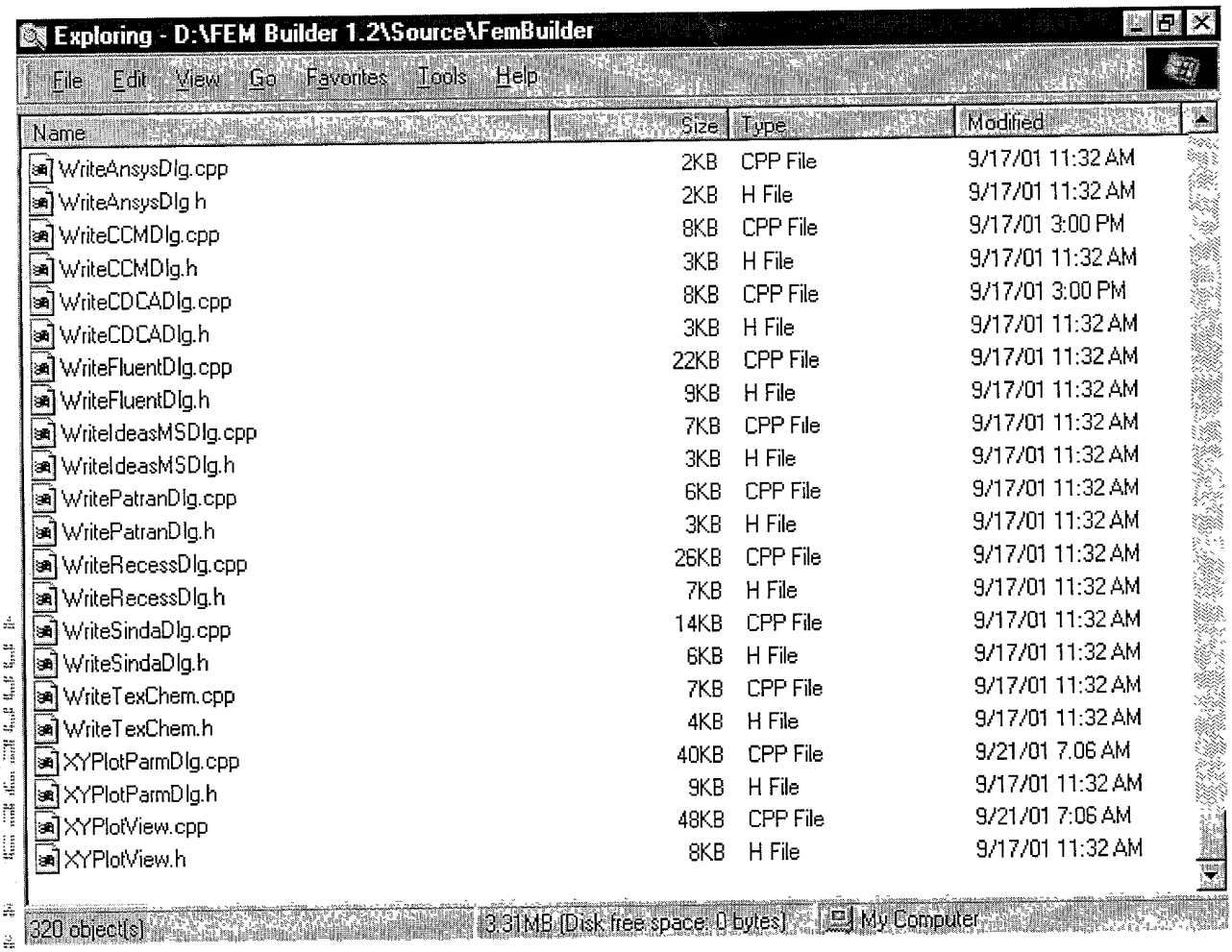

Exploring - D:\FEM Builder 1.2\Source\FemBuilder
File Edit View Go Favorites Tools Help
Name Size Type Modified
PostDeformedDlg.h 3KB H File 9/17/01 11:31 AM
PostDlg.cpp 33KB CPP File 9/21/01 7:06 AM
PostDlg.h 11KB H File 9/17/01 11:31 AM
PostVectorDlg.cpp 17KB CPP File 9/17/01 11:31 AM
PostVectorDlg.h 4KB H File 9/17/01 11:31 AM
PostView.cpp 15KB CPP File 9/17/01 11:31 AM
PostView.h 4KB H File 9/17/01 11:31 AM
PostXYAnalysisResultDlg.cpp 30KB CPP File 9/17/01 11:31 AM
PostXYAnalysisResultDlg.h 7KB H File 9/17/01 11:31 AM
PostXYPlotParmDlg.cpp 35KB CPP File 9/17/01 11:31 AM
PostXYPlotParmDlg.h 9KB H File 9/17/01 11:31 AM
PostXYPlotView.cpp 2KB CPP File 9/17/01 11:31 AM
PostXYPlotView.h 2KB H File 9/17/01 11:31 AM
PostXYTimeHistoryDlg.cpp 21KB CPP File 9/17/01 11:31 AM
PostXYTimeHistoryDlg.h 4KB H File 9/17/01 11:31 AM
ReadAbaqusDlg.cpp 3KB CPP File 9/17/01 11:31 AM
ReadAbaqusDlg.h 2KB H File 9/17/01 11:31 AM
ReadAnsysDlg.cpp 3KB CPP File 9/17/01 11:31 AM
ReadAnsysDlg.h 2KB H File 9/17/01 11:31 AM
ReadAscharDlg.cpp 4KB CPP File 9/17/01 11:31 AM
ReadAscharDlg.h 3KB H File 9/17/01 11:31 AM
ReadCCMDlg.cpp 4KB CPP File 9/17/01 11:31 AM
ReadCCMDlg.h 3KB H File 9/17/01 11:31 AM
320 object(s) 3.31MB (Disk free space: 0 bytes) My Computer

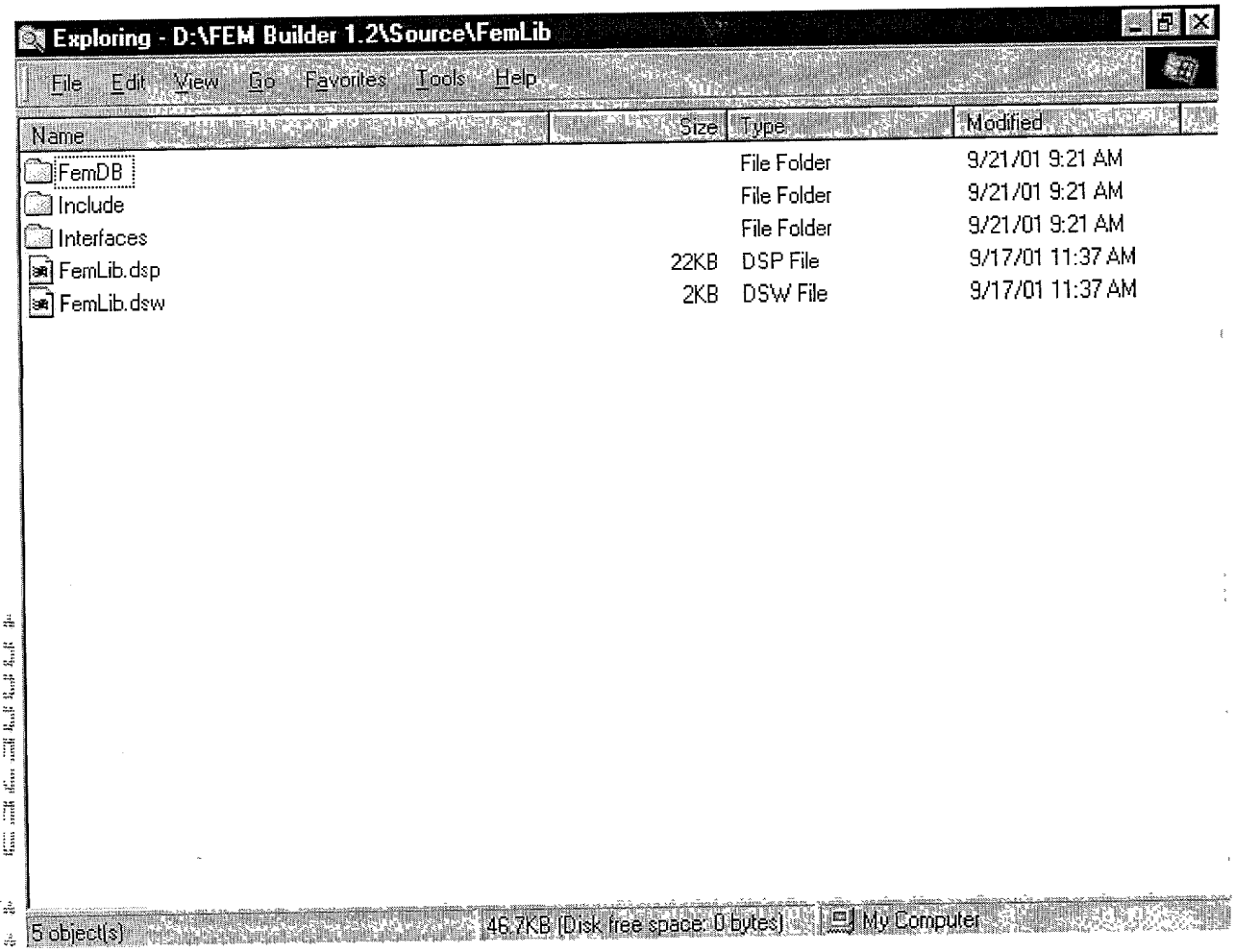

Exploring - D:\FEM Builder 1.2\Source\FemBuilder
File Edit View Go Favorites Tools Help
Name Size Type Modified
ReadCDCADlg.cpp 4KB CPP File 9/17/01 11:31 AM
ReadCDCADlg.h 3KB H File 9/17/01 11:31 AM
ReadFluentDlg.cpp 3KB CPP File 9/17/01 11:31 AM
ReadFluentDlg.h 2KB H File 9/17/01 11:31 AM
ReadGridGenDlg.cpp 3KB CPP File 9/17/01 11:31 AM
ReadGridGenDlg.h 2KB H File 9/17/01 11:31 AM
ReadIdeasMSDlg.cpp 3KB CPP File 9/17/01 11:31 AM
ReadIdeasMSDlg.h 2KB H File 9/17/01 11:31 AM
ReadPatranDlg.cpp 8KB CPP File 9/17/01 11:32 AM
ReadPatranDlg.h 3KB H File 9/17/01 11:32 AM
ReadRecessDlg.cpp 3KB CPP File 9/17/01 11:32 AM
ReadRecessDlg.h 2KB H File 9/17/01 11:32 AM
ReadSharpDlg.cpp 3KB CPP File 9/17/01 11:32 AM
ReadSharpDlg.h 2KB H File 9/17/01 11:32 AM
ReadSindaDlg.cpp 3KB CPP File 9/17/01 11:32 AM
ReadSindaDlg.h 2KB H File 9/17/01 11:32 AM
ReadSinGenDlg.cpp 3KB CPP File 9/17/01 11:32 AM
ReadSinGenDlg.h 2KB H File 9/17/01 11:32 AM
ReadTexChem.cpp 3KB CPP File 9/17/01 11:32 AM
ReadTexChem.h 2KB H File 9/17/01 11:32 AM
RefineMeshAdaptiveDlg.cpp 19KB CPP File 9/20/01 10:38 AM
RefineMeshAdaptiveDlg.h 5KB H File 9/17/01 11:32 AM
RefineMeshSpecifiedDlg.cpp 8KB CPP File 9/17/01 11:32 AM
320 object(s) 3.31MB (Disk free space: 0 bytes) My Computer

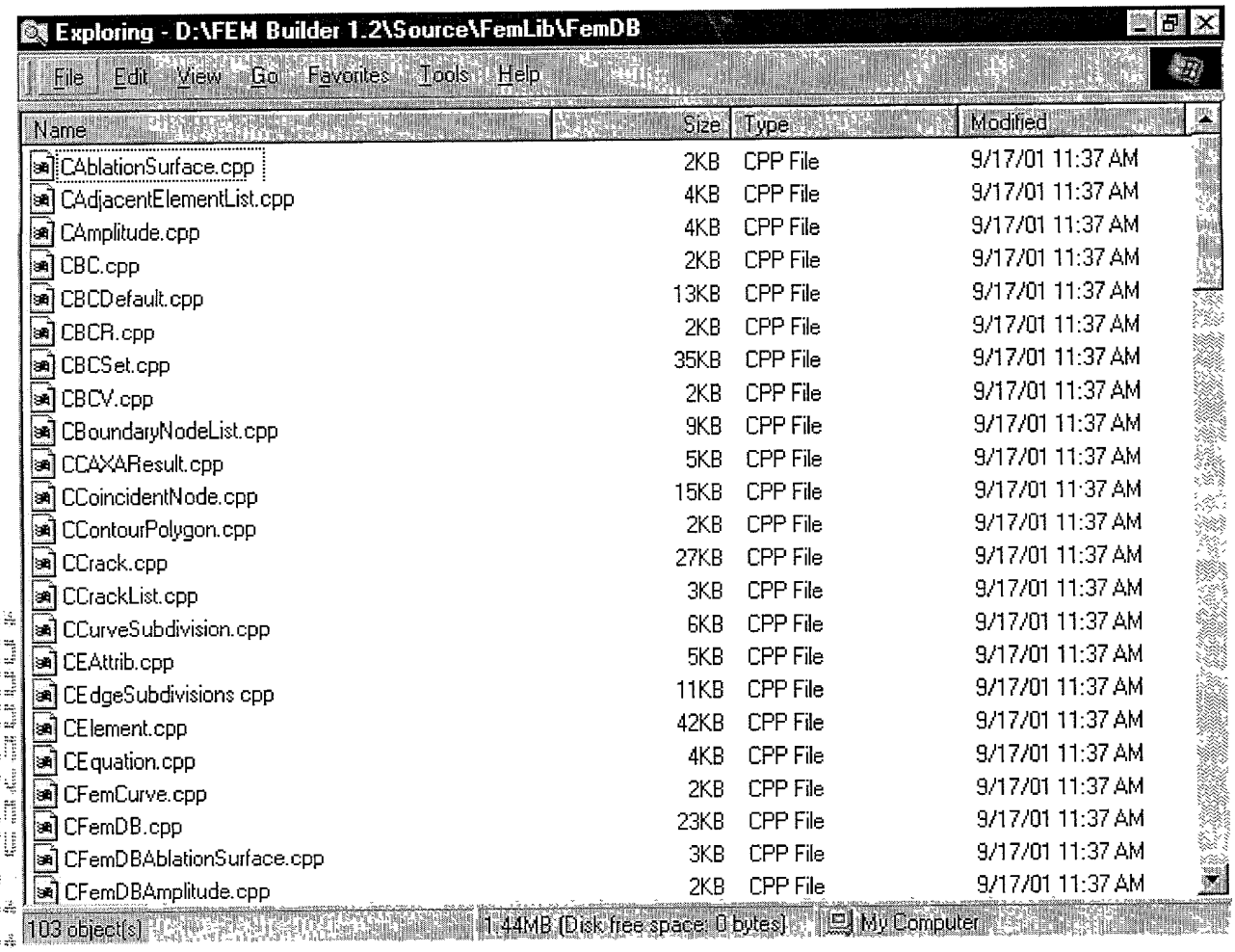

Exploring - D:\FEM Builder 1.2\Source\FemBuilder
File Edit View Go Favorites Tools Help
Name Size Type Modified
RefineMeshSpecifiedDlg.h 3KB H File 9/17/01 11:32 AM
resource.fd 56KB FD File 9/20/01 10:36 AM
resource.h 39KB H File 9/20/01 10:36 AM
ResultInterpolationDlg.cpp 13KB CPP File 9/17/01 11:32 AM
ResultInterpolationDlg.h 5KB H File 9/17/01 11:32 AM
ResultSuperpositionDlg.cpp 6KB CPP File 9/17/01 11:32 AM
ResultSuperpositionDlg.h 3KB H File 9/17/01 11:32 AM
SaveViewOrientationDlg.cpp 2KB CPP File 9/17/01 11:32 AM
SaveViewOrientationDlg.h 2KB H File 9/17/01 11:32 AM
SelectBCDlg.cpp 9KB CPP File 9/17/01 11:32 AM
SelectBCDlg.h 3KB H File 9/17/01 11:32 AM
SelectBCSetsDlg.cpp 3KB CPP File 9/17/01 11:32 AM
SelectBCSetsDlg.h 2KB H File 9/17/01 11:32 AM
SelectCurveDlg.cpp 9KB CPP File 9/17/01 11:32 AM
SelectCurveDlg.h 3KB H File 9/17/01 11:32 AM
SelectDlg.cpp 9KB CPP File 9/17/01 11:32 AM
SelectDlg.h 5KB H File 9/17/01 11:32 AM
SelectElementDlg.cpp 13KB CPP File 9/17/01 11:32 AM
SelectElementDlg.h 3KB H File 9/17/01 11:32 AM
SelectFaceDlg.cpp 8KB CPP File 9/17/01 11:32 AM
SelectFaceDlg.h 2KB H File 9/17/01 11:32 AM
SelectGroupDlg.cpp 8KB CPP File 9/17/01 11:32 AM
SelectGroupDlg.h 3KB H File 9/17/01 11:32 AM
320 object(s) 3.31 MB (Disk free space: 0 bytes) My Computer

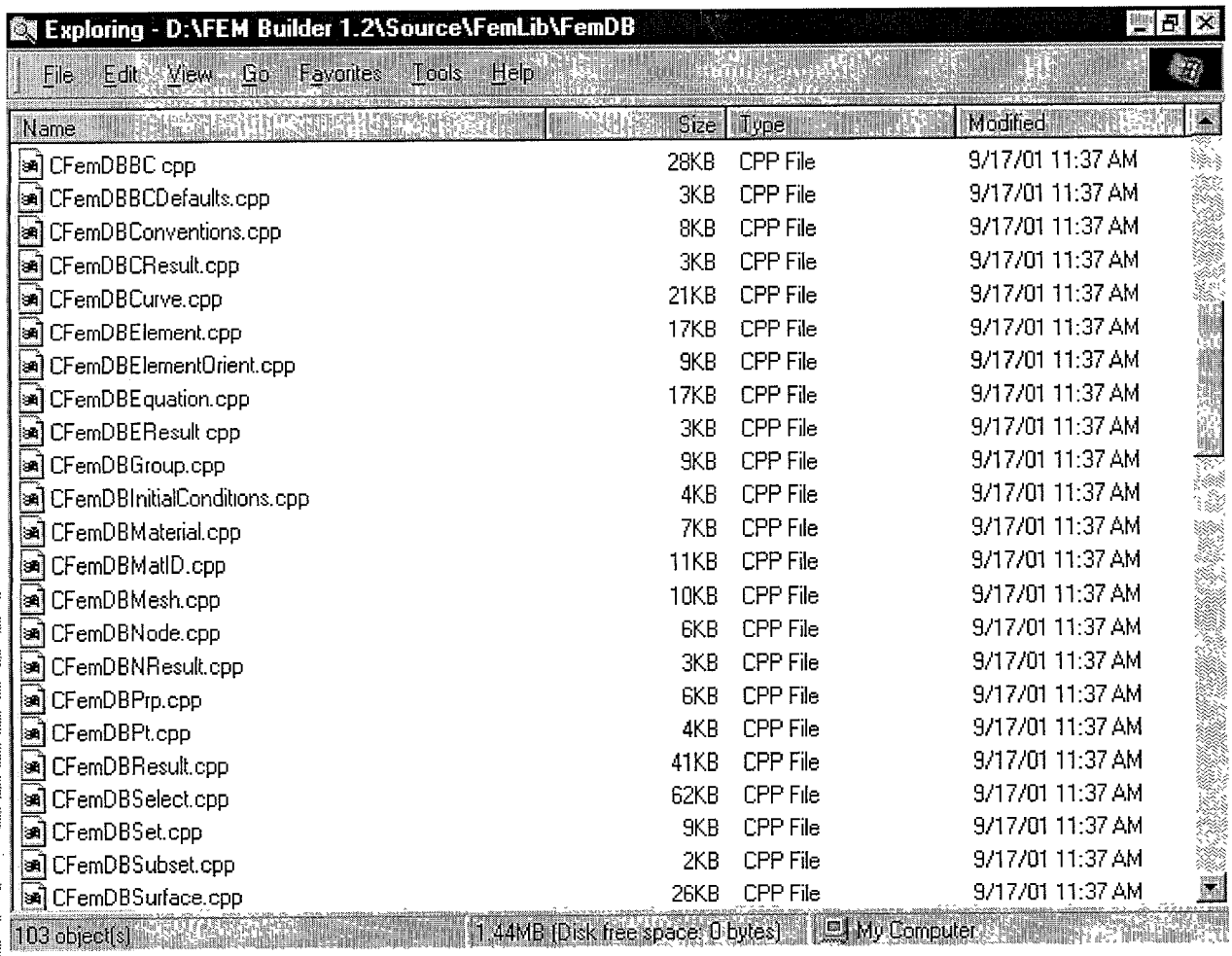

Exploring - D:\FEM Builder 1.2\Source\FemBuilder
File Edit View Go Favorites Tools Help
Name Size Type Modified
SelectMaterialDlg.cpp 8KB CPP File 9/17/01 11:32 AM
SelectMaterialDlg.h 3KB H File 9/17/01 11:32 AM
SelectMaterialIDsDlg.cpp 3KB CPP File 9/17/01 11:32 AM
SelectMaterialIDsDlg.h 2KB H File 9/17/01 11:32 AM
SelectModelDlg.cpp 3KB CPP File 9/17/01 11:32 AM
SelectModelDlg.h 2KB H File 9/17/01 11:32 AM
SelectNodeDlg.cpp 13KB CPP File 9/17/01 11:32 AM
SelectNodeDlg.h 3KB H File 9/17/01 11:32 AM
SelectPointDlg.cpp 9KB CPP File 9/17/01 11:32 AM
SelectPointDlg.h 3KB H File 9/17/01 11:32 AM
SelectReactionsDlg.cpp 4KB CPP File 9/17/01 11:32 AM
SelectReactionsDlg.h 2KB H File 9/17/01 11:32 AM
SelectResultDlg.cpp 16KB CPP File 9/17/01 11:32 AM
SelectResultDlg.h 4KB H File 9/17/01 11:32 AM
SelectSpeciesDlg.cpp 4KB CPP File 9/17/01 11:32 AM
SelectSpeciesDlg.h 2KB H File 9/17/01 11:32 AM
SelectSurfaceByTypeDlg.cpp 3KB CPP File 9/17/01 11:32 AM
SelectSurfaceByTypeDlg.h 2KB H File 9/17/01 11:32 AM
SelectSurfaceDlg.cpp 10KB CPP File 9/17/01 11:32 AM
SelectSurfaceDlg.h 3KB H File 9/17/01 11:32 AM
SelectSystemDlg.cpp 6KB CPP File 9/17/01 11:32 AM
SelectSystemDlg.h 2KB H File 9/17/01 11:32 AM
SelectVolumeDlg.cpp 9KB CPP File 9/17/01 11:32 AM
320 object(s) 3.31 MB (Disk free space: 0 bytes) My Computer

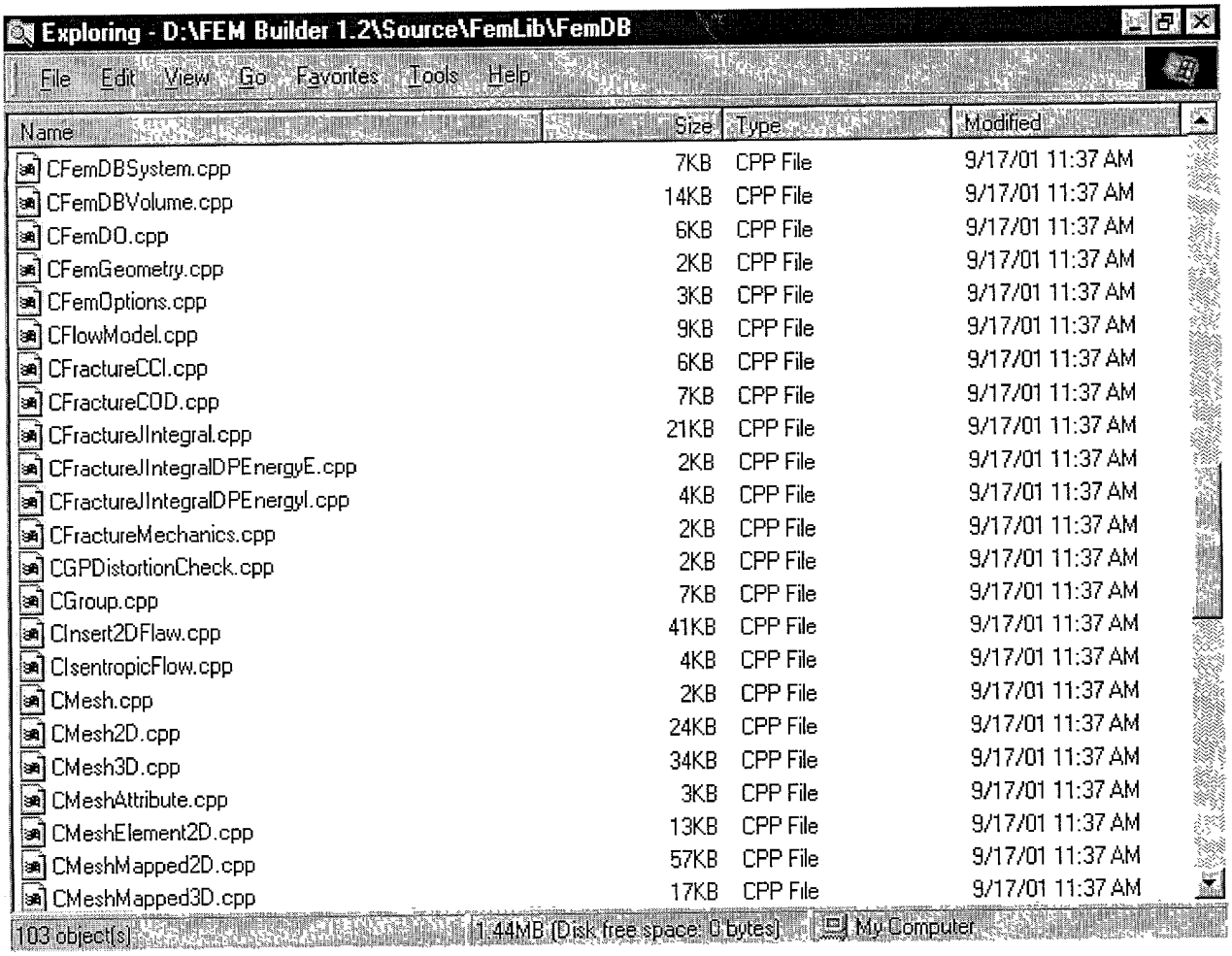

Exploring - D:\FEM Builder 1.2\Source\FemBuilder
File Edit View Go Favorites Tools Help
Name Size Type Modified
SelectVolumeDlg.h 3KB H File 9/17/01 11:32 AM
SmoothMeshDlg.cpp 7KB CPP File 9/17/01 11:32 AM
SmoothMeshDlg.h 3KB H File 9/17/01 11:32 AM
SplashWnd.cpp 4KB CPP File 9/17/01 11:32 AM
SplashWnd.h 2KB H File 9/17/01 11:32 AM
Splsh16.bmp 81KB Bitmap Image 9/17/01 11:32 AM
StdAfx.cpp 1KB CPP File 9/17/01 11:32 AM
StdAfx.h 2KB H File 9/17/01 11:32 AM
SubModelDlg.cpp 11KB CPP File 9/17/01 11:32 AM
SubModelDlg.h 4KB H File 9/17/01 11:32 AM
SweepDlg.cpp 8KB CPP File 9/17/01 11:32 AM
SweepDlg.h 3KB H File 9/17/01 11:32 AM
ThermalAblationDlg.cpp 12KB CPP File 9/17/01 11:32 AM
ThermalAblationDlg.h 4KB H File 9/17/01 11:32 AM
TranslateDlg.cpp 2KB CPP File 9/17/01 11:32 AM
TranslateDlg.h 2KB H File 9/17/01 11:32 AM
UnitSystemPP.cpp 6KB CPP File 9/17/01 11:32 AM
UnitSystemPP.h 2KB H File 9/17/01 11:32 AM
WriteAbaqusDlg.cpp 18KB CPP File 9/17/01 11:32 AM
WriteAbaqusDlg.h 7KB H File 9/17/01 11:32 AM
WriteAnsysDlg.cpp 2KB CPP File 9/17/01 11:32 AM
WriteAnsysDlg.h 2KB H File 9/17/01 11:32 AM
WriteCCMDlg.cpp 8KB CPP File 9/17/01 3:00 PM
320 object(s) 3.31MB (Disk free space: 0 bytes) My Computer

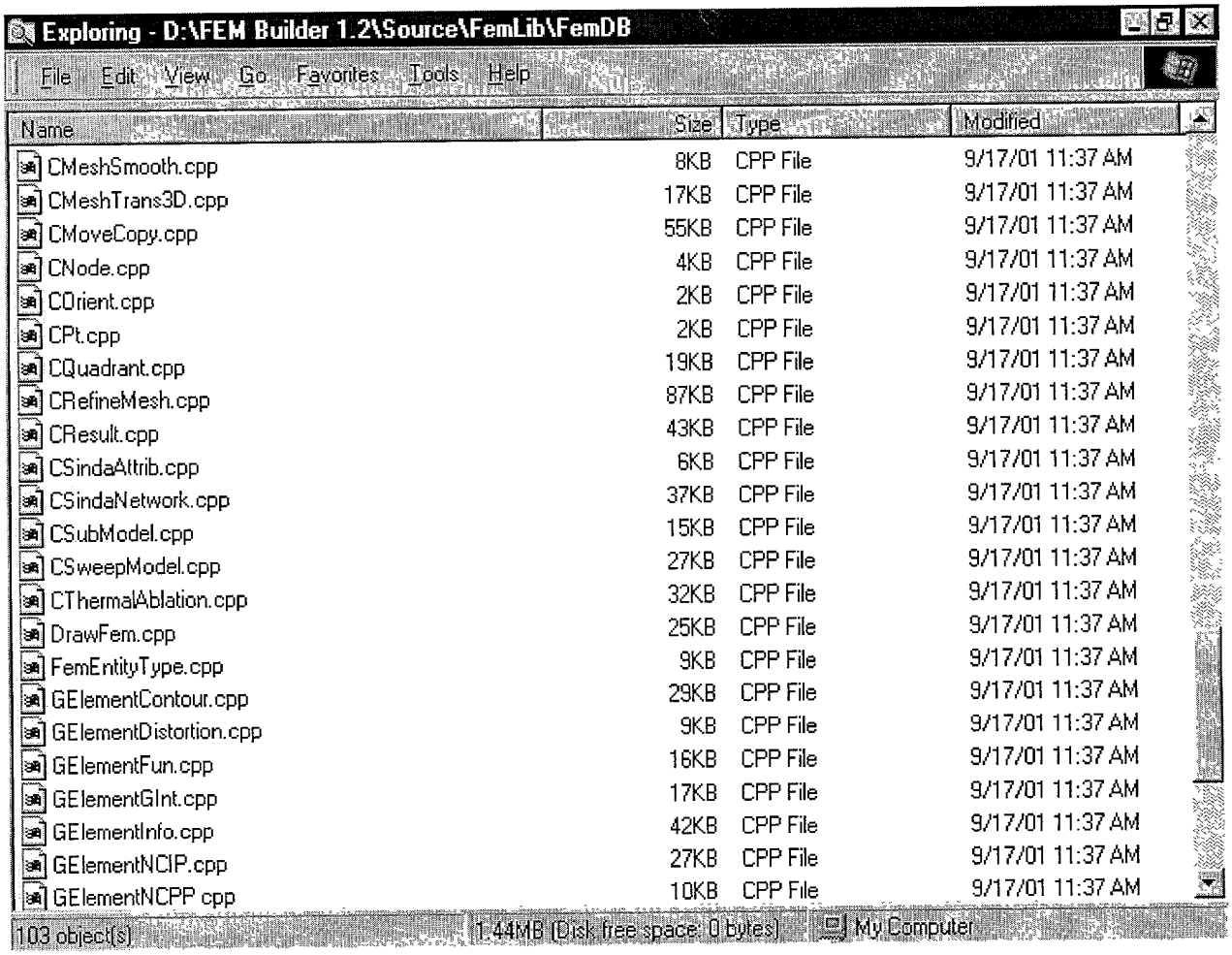

Exploring - D:\FEM Builder 1.2\Source\FemBuilder
File Edit View Go Favorites Tools Help
Name Size Type Modified
WriteAnsysDlg.cpp 2KB CPP File 9/17/01 11:32 AM
WriteAnsysDlg.h 2KB H File 9/17/01 11:32 AM
WriteCCMDlg.cpp 8KB CPP File 9/17/01 3:00 PM
WriteCCMDlg.h 3KB H File 9/17/01 11:32 AM
WriteCDCADlg.cpp 8KB CPP File 9/17/01 3:00 PM
WriteCDCADlg.h 3KB H File 9/17/01 11:32 AM
WriteFluentDlg.cpp 22KB CPP File 9/17/01 11:32 AM
WriteFluentDlg.h 9KB H File 9/17/01 11:32 AM
WriteIdeasMSDlg.cpp 7KB CPP File 9/17/01 11:32 AM
WriteIdeasMSDlg.h 3KB H File 9/17/01 11:32 AM
WritePatranDlg.cpp 6KB CPP File 9/17/01 11:32 AM
WritePatranDlg.h 3KB H File 9/17/01 11:32 AM
WriteRecessDlg.cpp 26KB CPP File 9/17/01 11:32 AM
WriteRecessDlg.h 7KB H File 9/17/01 11:32 AM
WriteSindaDlg.cpp 14KB CPP File 9/17/01 11:32 AM
WriteSindaDlg.h 6KB H File 9/17/01 11:32 AM
WriteTexChem.cpp 7KB CPP File 9/17/01 11:32 AM
WriteTexChem.h 4KB H File 9/17/01 11:32 AM
XYPlotParmDlg.cpp 40KB CPP File 9/21/01 7:06 AM
XYPlotParmDlg.h 9KB H File 9/17/01 11:32 AM
XYPlotView.cpp 48KB CPP File 9/21/01 7:06 AM
XYPlotView.h 8KB H File 9/17/01 11:32 AM
320 object(s) 3.31 MB (Disk free space: 0 bytes) My Computer

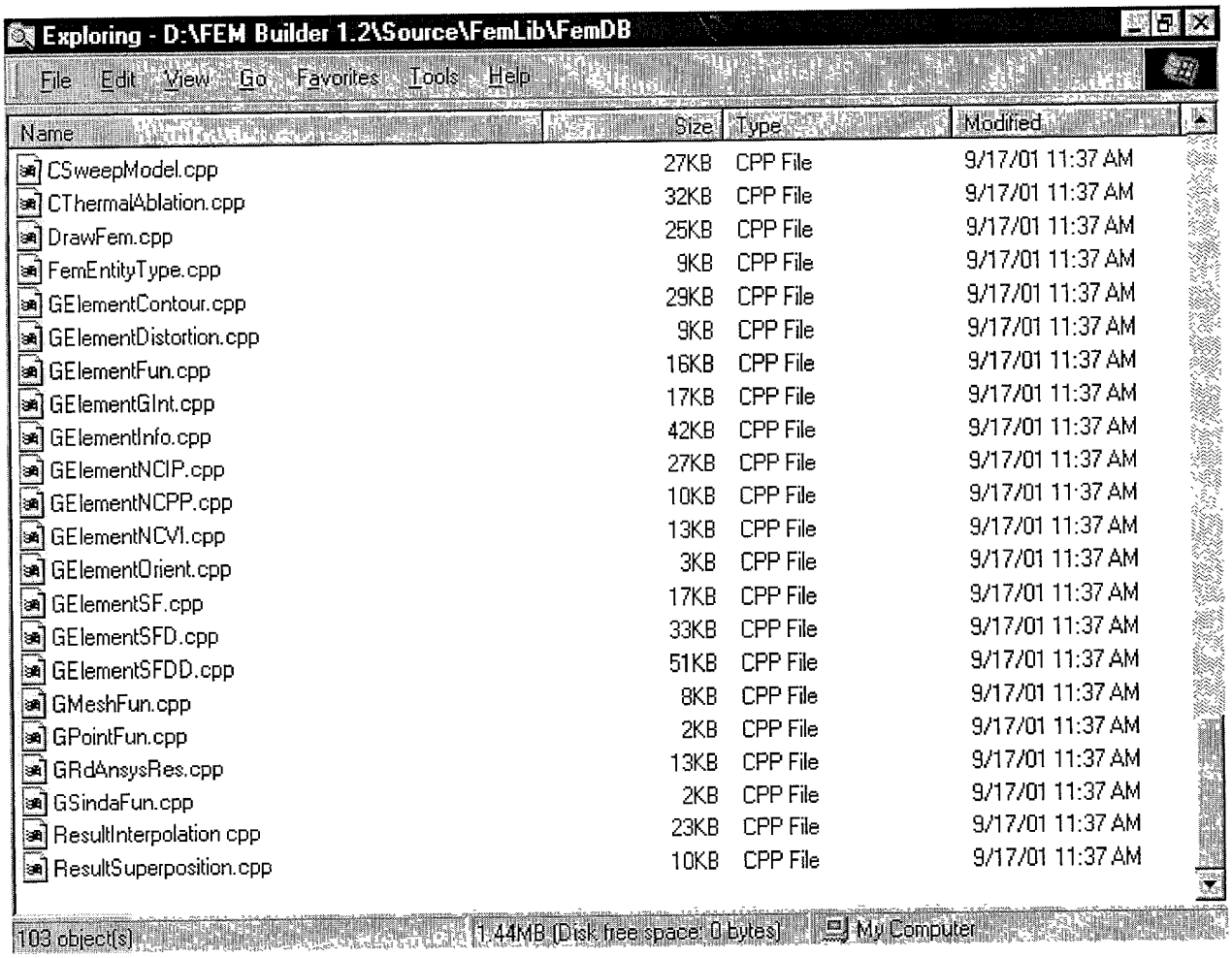

Exploring - D:\FEM Builder 1.2\Source\FemLib
File Edit View Go Favorites Tools Help
Name Size Type Modified
FemDB File Folder 9/21/01 9:21 AM
Include File Folder 9/21/01 9:21 AM
Interfaces File Folder 9/21/01 9:21 AM
FemLib.dsp 22KB DSP File 9/17/01 11:37 AM
FemLib.dsw 2KB DSW File 9/17/01 11:37 AM
5 object(s)
46.7KB (Disk free space: 0 bytes)
My Computer

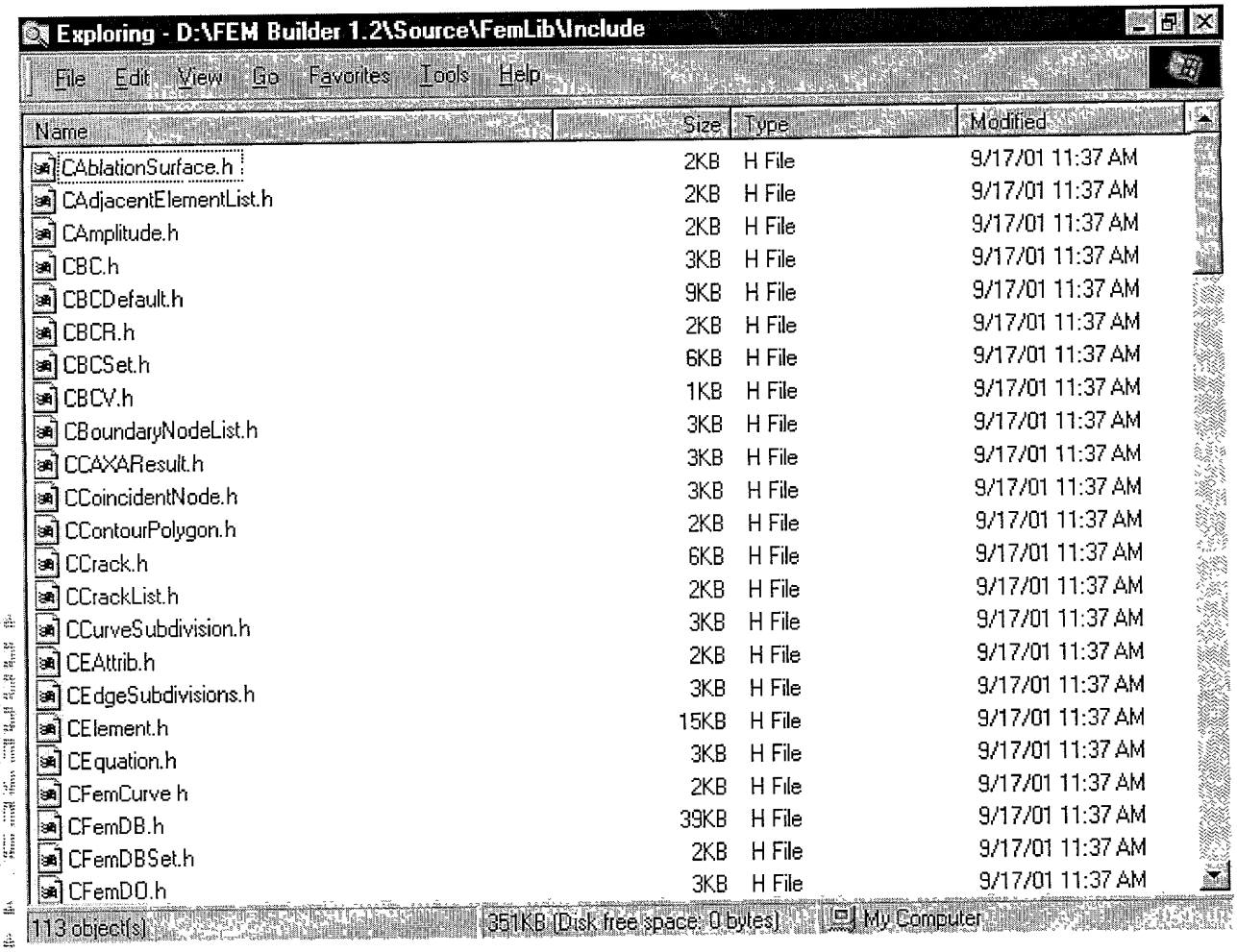

Exploring - D:\FEM Builder 1.2\Source\FemLib\FemDB
File Edit View Go Favorites Tools Help
Name Size Type Modified
CAblationSurface.cpp 2KB CPP File 9/17/01 11:37 AM
CAdjacentElementList.cpp 4KB CPP File 9/17/01 11:37 AM
CAmplitude.cpp 4KB CPP File 9/17/01 11:37 AM
CBC.cpp 2KB CPP File 9/17/01 11:37 AM
CBCDefault.cpp 13KB CPP File 9/17/01 11:37 AM
CBCR.cpp 2KB CPP File 9/17/01 11:37 AM
CBCSet.cpp 35KB CPP File 9/17/01 11:37 AM
CBCV.cpp 2KB CPP File 9/17/01 11:37 AM
CBoundaryNodeList.cpp 9KB CPP File 9/17/01 11:37 AM
CCAXAResult.cpp 5KB CPP File 9/17/01 11:37 AM
CCoincidentNode.cpp 15KB CPP File 9/17/01 11:37 AM
CContourPolygon.cpp 2KB CPP File 9/17/01 11:37 AM
CCrack.cpp 27KB CPP File 9/17/01 11:37 AM
CCrackList.cpp 3KB CPP File 9/17/01 11:37 AM
CCurveSubdivision.cpp 6KB CPP File 9/17/01 11:37 AM
CEAttrib.cpp 5KB CPP File 9/17/01 11:37 AM
CEdgeSubdivisions.cpp 11KB CPP File 9/17/01 11:37 AM
CElement.cpp 42KB CPP File 9/17/01 11:37 AM
CEquation.cpp 4KB CPP File 9/17/01 11:37 AM
CFemCurve.cpp 2KB CPP File 9/17/01 11:37 AM
CFemDB.cpp 23KB CPP File 9/17/01 11:37 AM
CFemDBAblationSurface.cpp 3KB CPP File 9/17/01 11:37 AM
CFemDBAmplitude.cpp 2KB CPP File 9/17/01 11:37 AM
103 object(s) 1.44MB (Disk free space: 0 bytes) My Computer

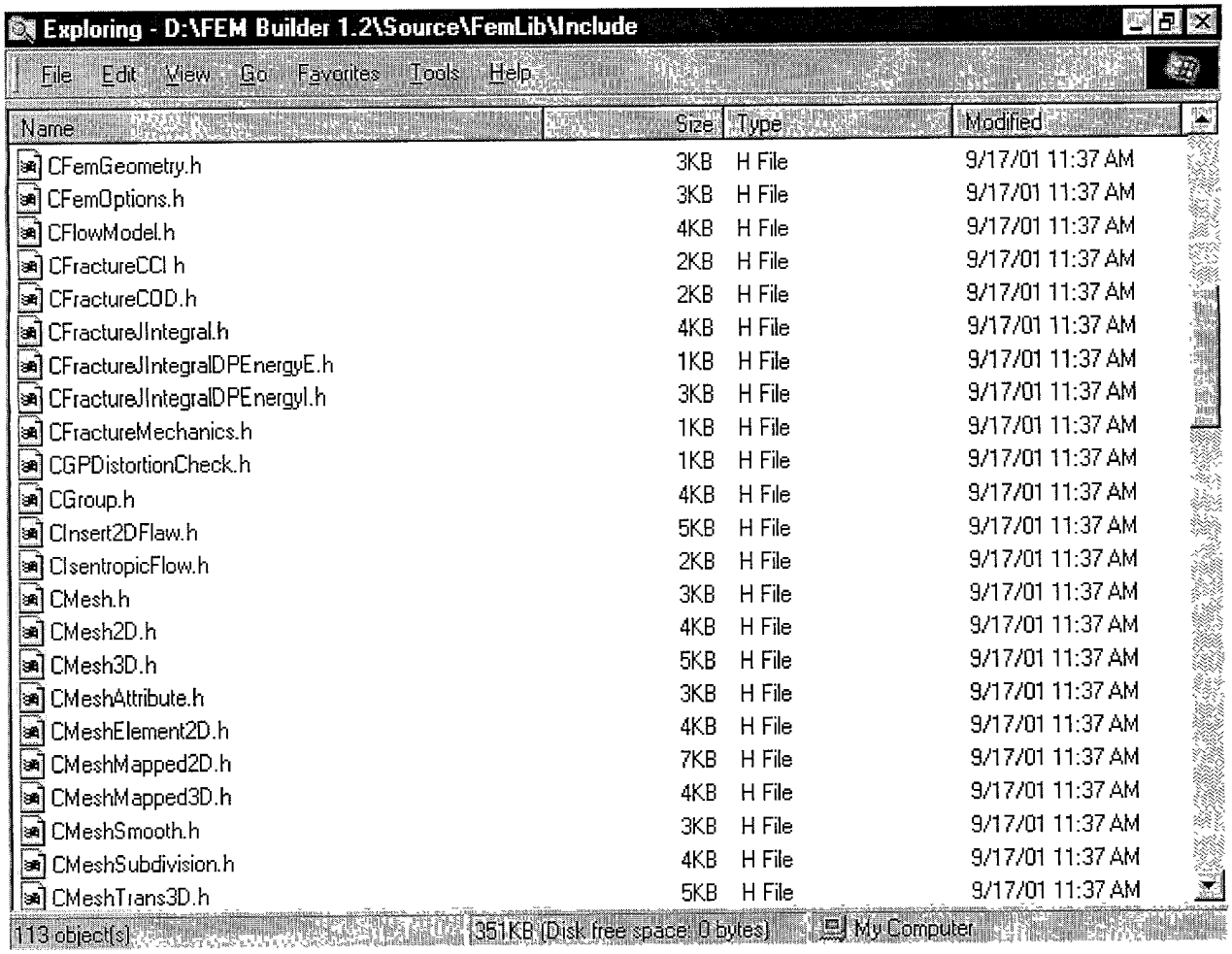

Exploring - D:\FEM Builder 1.2\Source\FemLib\FemDB
File Edit View Go Favorites Tools Help
Name Size Type Modified
CFemDBBC.cpp 28KB CPP File 9/17/01 11:37 AM
CFemDBBCDefaults.cpp 3KB CPP File 9/17/01 11:37 AM
CFemDBConventions.cpp 8KB CPP File 9/17/01 11:37 AM
CFemDBCResult.cpp 3KB CPP File 9/17/01 11:37 AM
CFemDBCurve.cpp 21KB CPP File 9/17/01 11:37 AM
CFemDBElement.cpp 17KB CPP File 9/17/01 11:37 AM
CFemDBElementOrient.cpp 9KB CPP File 9/17/01 11:37 AM
CFemDBEquation.cpp 17KB CPP File 9/17/01 11:37 AM
CFemDBEResult.cpp 3KB CPP File 9/17/01 11:37 AM
CFemDBGroup.cpp 9KB CPP File 9/17/01 11:37 AM
CFemDBInitialConditions.cpp 4KB CPP File 9/17/01 11:37 AM
CFemDBMaterial.cpp 7KB CPP File 9/17/01 11:37 AM
CFemDBMatID.cpp 11KB CPP File 9/17/01 11:37 AM
CFemDBMesh.cpp 10KB CPP File 9/17/01 11:37 AM
CFemDBNode.cpp 6KB CPP File 9/17/01 11:37 AM
CFemDBNResult.cpp 3KB CPP File 9/17/01 11:37 AM
CFemDBPrp.cpp 6KB CPP File 9/17/01 11:37 AM
CFemDBPt.cpp 4KB CPP File 9/17/01 11:37 AM
CFemDBResult.cpp 41KB CPP File 9/17/01 11:37 AM
CFemDBSelect.cpp 62KB CPP File 9/17/01 11:37 AM
CFemDBSet.cpp 9KB CPP File 9/17/01 11:37 AM
CFemDBSubset.cpp 2KB CPP File 9/17/01 11:37 AM
CFemDBSurface.cpp 26KB CPP File 9/17/01 11:37 AM
103 object(s) 1.44MB (Disk free space: 0 bytes) My Computer

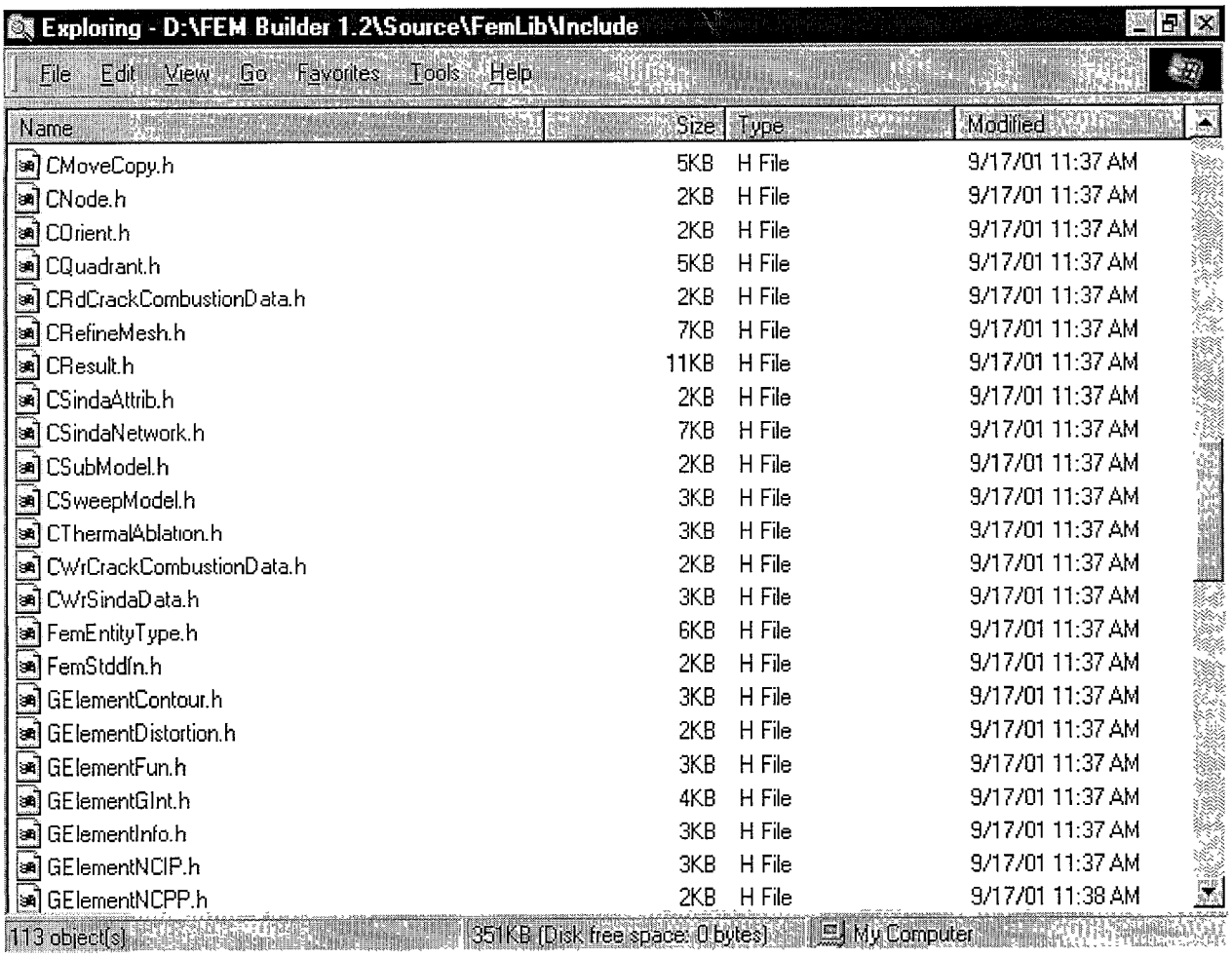

Exploring - D:\FEM Builder 1.2\Source\FemLib\FemDB
File Edit View Go Favorites Tools Help
Name Size Type Modified
CFemDBSystem.cpp 7KB CPP File 9/17/01 11:37 AM
CFemDBVolume.cpp 14KB CPP File 9/17/01 11:37 AM
CFemDO.cpp 6KB CPP File 9/17/01 11:37 AM
CFemGeometry.cpp 2KB CPP File 9/17/01 11:37 AM
CFemOptions.cpp 3KB CPP File 9/17/01 11:37 AM
CFlowModel.cpp 9KB CPP File 9/17/01 11:37 AM
CFractureCCI.cpp 6KB CPP File 9/17/01 11:37 AM
CFractureCOD.cpp 7KB CPP File 9/17/01 11:37 AM
CFractureJIntegral.cpp 21KB CPP File 9/17/01 11:37 AM
CFractureJIntegralDPEnergyE.cpp 2KB CPP File 9/17/01 11:37 AM
CFractureJIntegralDPEnergyI.cpp 4KB CPP File 9/17/01 11:37 AM
CFractureMechanics.cpp 2KB CPP File 9/17/01 11:37 AM
CGPDistortionCheck.cpp 2KB CPP File 9/17/01 11:37 AM
CGroup.cpp 7KB CPP File 9/17/01 11:37 AM
CInsert2DFlaw.cpp 41KB CPP File 9/17/01 11:37 AM
CIsentropicFlow.cpp 4KB CPP File 9/17/01 11:37 AM
CMesh.cpp 2KB CPP File 9/17/01 11:37 AM
CMesh2D.cpp 24KB CPP File 9/17/01 11:37 AM
CMesh3D.cpp 34KB CPP File 9/17/01 11:37 AM
CMeshAttribute.cpp 3KB CPP File 9/17/01 11:37 AM
CMeshElement2D.cpp 13KB CPP File 9/17/01 11:37 AM
CMeshMapped2D.cpp 57KB CPP File 9/17/01 11:37 AM
CMeshMapped3D.cpp 17KB CPP File 9/17/01 11:37 AM
103 object(s) 1.44MB (Disk free space: 0 bytes) My Computer

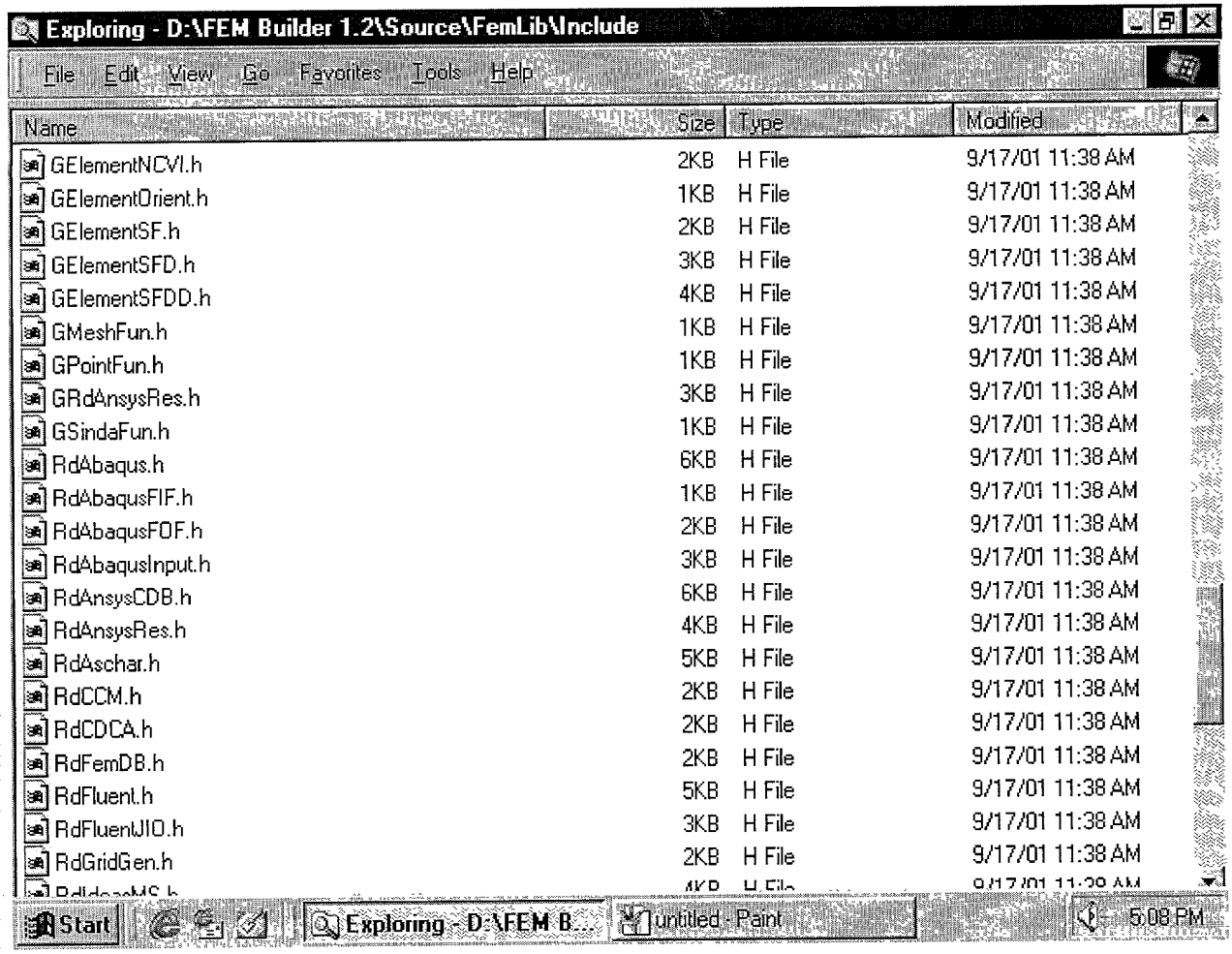

Exploring - D:\FEM Builder 1.2\Source\FemLib\FemDB
File Edit View Go Favorites Tools Help
Name Size Type Modified
CMeshSmooth.cpp 8KB CPP File 9/17/01 11:37 AM
CMeshTrans3D.cpp 17KB CPP File 9/17/01 11:37 AM
CMoveCopy.cpp 55KB CPP File 9/17/01 11:37 AM
CNode.cpp 4KB CPP File 9/17/01 11:37 AM
COrient.cpp 2KB CPP File 9/17/01 11:37 AM
CPt.cpp 2KB CPP File 9/17/01 11:37 AM
CQuadrant.cpp 19KB CPP File 9/17/01 11:37 AM
CRefineMesh.cpp 87KB CPP File 9/17/01 11:37 AM
CResult.cpp 43KB CPP File 9/17/01 11:37 AM
CSindaAttrib.cpp 6KB CPP File 9/17/01 11:37 AM
CSindaNetwork.cpp 37KB CPP File 9/17/01 11:37 AM
CSubModel.cpp 15KB CPP File 9/17/01 11:37 AM
CSweepModel.cpp 27KB CPP File 9/17/01 11:37 AM
CThermalAblation.cpp 32KB CPP File 9/17/01 11:37 AM
DrawFem.cpp 25KB CPP File 9/17/01 11:37 AM
FemEntityType.cpp 9KB CPP File 9/17/01 11:37 AM
GElementContour.cpp 29KB CPP File 9/17/01 11:37 AM
GElementDistortion.cpp 9KB CPP File 9/17/01 11:37 AM
GElementFun.cpp 16KB CPP File 9/17/01 11:37 AM
GElementGInt.cpp 17KB CPP File 9/17/01 11:37 AM
GElementInfo.cpp 42KB CPP File 9/17/01 11:37 AM
GElementNCIP.cpp 27KB CPP File 9/17/01 11:37 AM
GElementNCPP.cpp 10KB CPP File 9/17/01 11:37 AM
103 object(s) 1.44MB (Disk free space: 0 bytes) My Computer

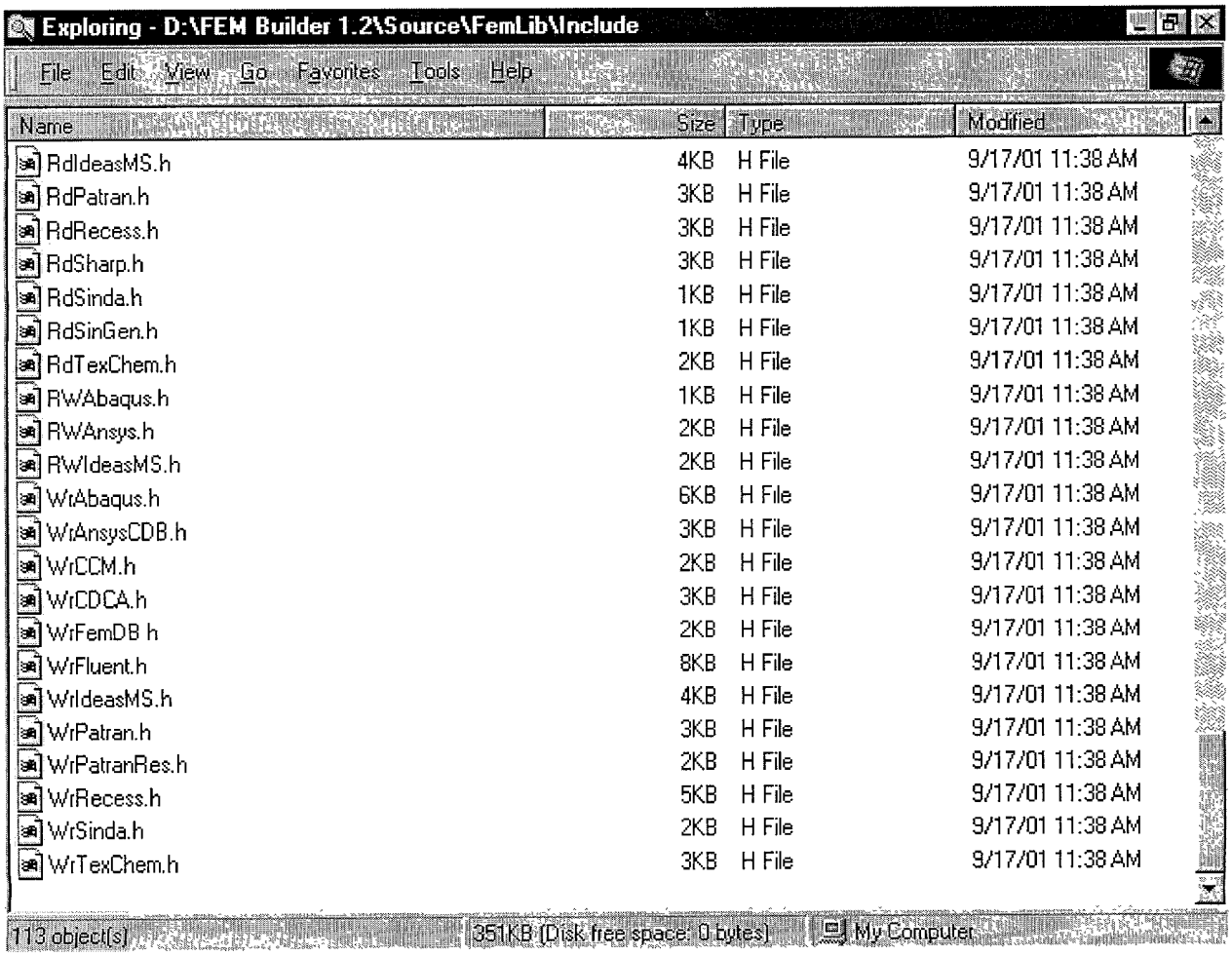

Exploring - D:\FEM Builder 1.2\Source\FemLib\FemDB
File Edit View Go Favorites Tools Help
Name Size Type Modified
CSweepModel.cpp 27KB CPP File 9/17/01 11:37 AM
CThermalAblation.cpp 32KB CPP File 9/17/01 11:37 AM
DrawFem.cpp 25KB CPP File 9/17/01 11:37 AM
FemEntityType.cpp 9KB CPP File 9/17/01 11:37 AM
GElementContour.cpp 29KB CPP File 9/17/01 11:37 AM
GElementDistortion.cpp 9KB CPP File 9/17/01 11:37 AM
GElementFun.cpp 16KB CPP File 9/17/01 11:37 AM
GElementGInt.cpp 17KB CPP File 9/17/01 11:37 AM
GElementInfo.cpp 42KB CPP File 9/17/01 11:37 AM
GElementNCIP.cpp 27KB CPP File 9/17/01 11:37 AM
GElementNCPP.cpp 10KB CPP File 9/17/01 11:37 AM
GElementNCVI.cpp 13KB CPP File 9/17/01 11:37 AM
GElementOrient.cpp 3KB CPP File 9/17/01 11:37 AM
GElementSF.cpp 17KB CPP File 9/17/01 11:37 AM
GElementSFD.cpp 33KB CPP File 9/17/01 11:37 AM
GElementSFDD.cpp 51KB CPP File 9/17/01 11:37 AM
GMeshFun.cpp 8KB CPP File 9/17/01 11:37 AM
GPointFun.cpp 2KB CPP File 9/17/01 11:37 AM
GRdAnsysRes.cpp 13KB CPP File 9/17/01 11:37 AM
GSindaFun.cpp 2KB CPP File 9/17/01 11:37 AM
ResultInterpolation.cpp 23KB CPP File 9/17/01 11:37 AM
ResultSuperposition.cpp 10KB CPP File 9/17/01 11:37 AM
103 object(s) 1.44MB (Disk free space: 0 bytes) My Computer

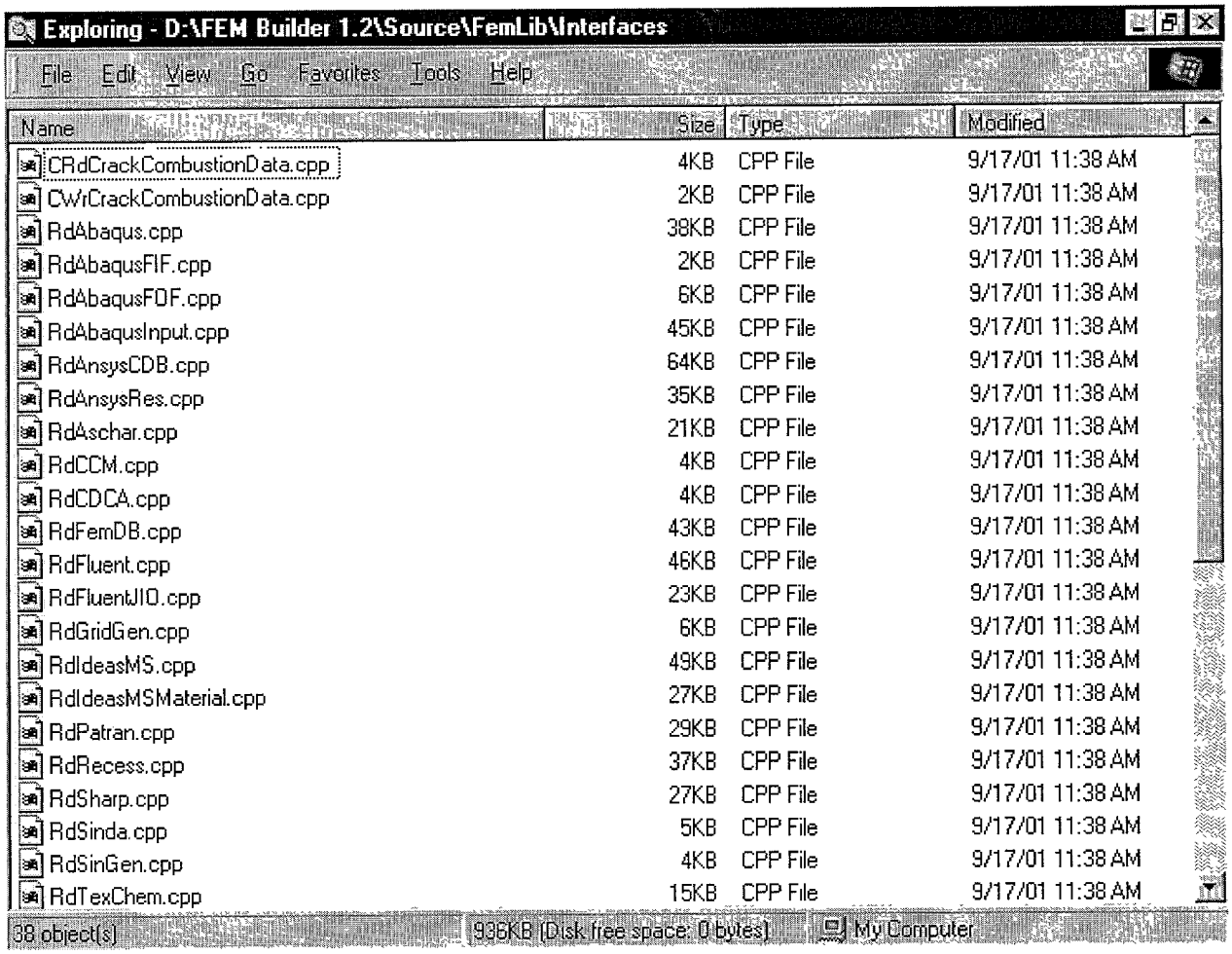

Exploring - D:\FEM Builder 1.2\Source\FemLib\Include
File Edit View Go Favorites Tools Help
Name Size Type Modified
CAblationSurface.h 2KB H File 9/17/01 11:37 AM
CAdjacentElementList.h 2KB H File 9/17/01 11:37 AM
CAmplitude.h 2KB H File 9/17/01 11:37 AM
CBC.h 3KB H File 9/17/01 11:37 AM
CBCDefault.h 9KB H File 9/17/01 11:37 AM
CBCR.h 2KB H File 9/17/01 11:37 AM
CBCSet.h 6KB H File 9/17/01 11:37 AM
CBCV.h 1KB H File 9/17/01 11:37 AM
CBoundaryNodeList.h 3KB H File 9/17/01 11:37 AM
CCAXAResult.h 3KB H File 9/17/01 11:37 AM
CCoincidentNode.h 3KB H File 9/17/01 11:37 AM
CContourPolygon.h 2KB H File 9/17/01 11:37 AM
CCrack.h 6KB H File 9/17/01 11:37 AM
CCrackList.h 2KB H File 9/17/01 11:37 AM
CCurveSubdivision.h 3KB H File 9/17/01 11:37 AM
CEAttrib.h 2KB H File 9/17/01 11:37 AM
CEdgeSubdivisions.h 3KB H File 9/17/01 11:37 AM
CElement.h 15KB H File 9/17/01 11:37 AM
CEquation.h 3KB H File 9/17/01 11:37 AM
CFemCurve.h 2KB H File 9/17/01 11:37 AM
CFemDB.h 39KB H File 9/17/01 11:37 AM
CFemDBSet.h 2KB H File 9/17/01 11:37 AM
CFemDO.h 3KB H File 9/17/01 11:37 AM
113 object(s) 351KB (Disk free space: 0 bytes) My Computer

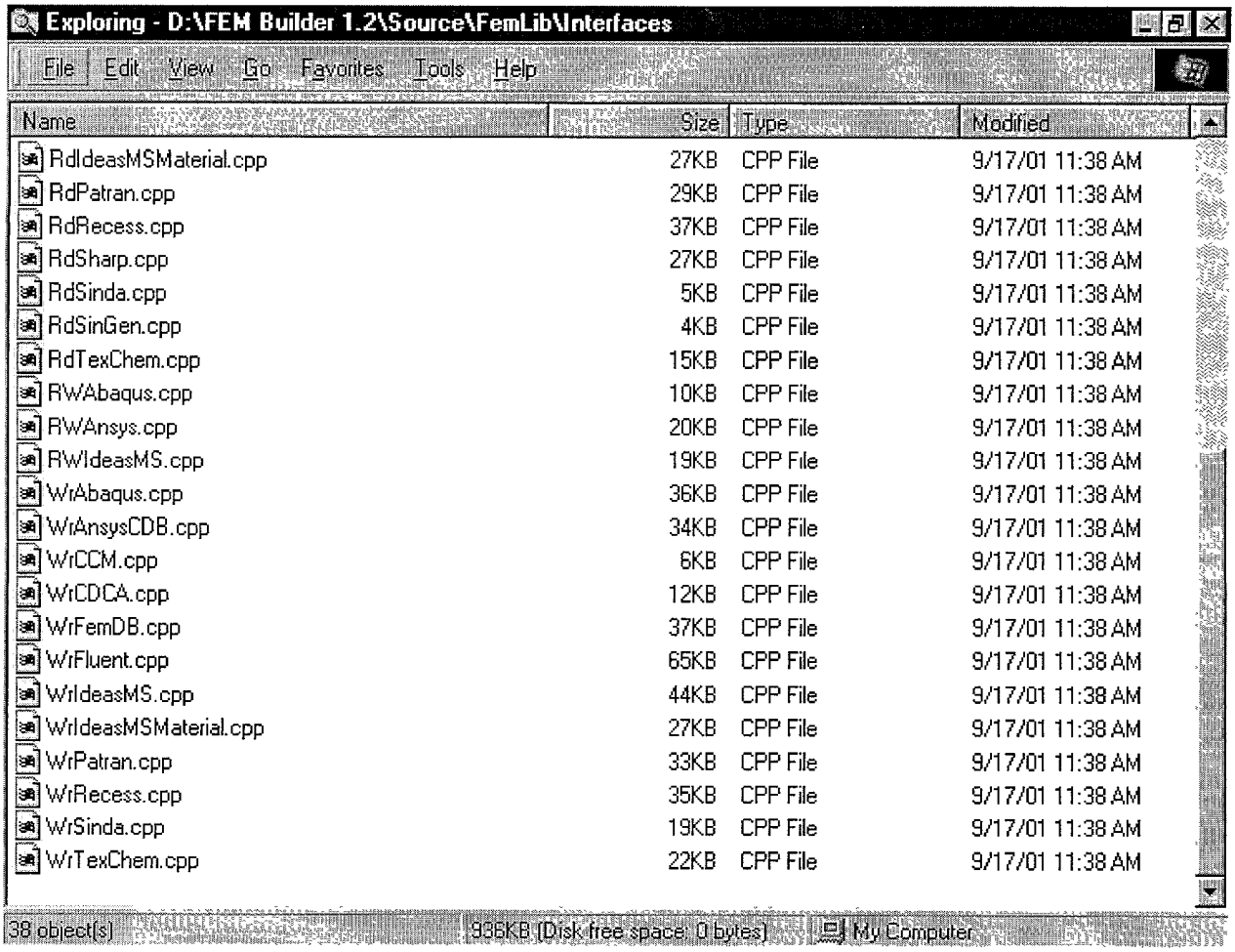

Exploring - D:\FEM Builder 1.2\Source\FemLib\Include
File Edit View Go Favorites Tools Help
Name Size Type Modified
CFemGeometry.h 3KB H File 9/17/01 11:37 AM
CFemOptions.h 3KB H File 9/17/01 11:37 AM
CFlowModel.h 4KB H File 9/17/01 11:37 AM
CFractureCCI h 2KB H File 9/17/01 11:37 AM
CFractureCOD.h 2KB H File 9/17/01 11:37 AM
CFractureJIntegral.h 4KB H File 9/17/01 11:37 AM
CFractureJIntegralDPEnergyE.h 1KB H File 9/17/01 11:37 AM
CFractureJIntegralDPEnergyI.h 3KB H File 9/17/01 11:37 AM
CFractureMechanics.h 1KB H File 9/17/01 11:37 AM
CGPDistortionCheck.h 1KB H File 9/17/01 11:37 AM
CGroup.h 4KB H File 9/17/01 11:37 AM
CInsert2DFlaw.h 5KB H File 9/17/01 11:37 AM
CIsentropicFlow.h 2KB H File 9/17/01 11:37 AM
CMesh.h 3KB H File 9/17/01 11:37 AM
CMesh2D.h 4KB H File 9/17/01 11:37 AM
CMesh3D.h 5KB H File 9/17/01 11:37 AM
CMeshAttribute.h 3KB H File 9/17/01 11:37 AM
CMeshElement2D.h 4KB H File 9/17/01 11:37 AM
CMeshMapped2D.h 7KB H File 9/17/01 11:37 AM
CMeshMapped3D.h 4KB H File 9/17/01 11:37 AM
CMeshSmooth.h 3KB H File 9/17/01 11:37 AM
CMeshSubdivision.h 4KB H File 9/17/01 11:37 AM
CMeshTrans3D.h 5KB H File 9/17/01 11:37 AM
113 object(s) 351KB (Disk free space: 0 bytes) My Computer

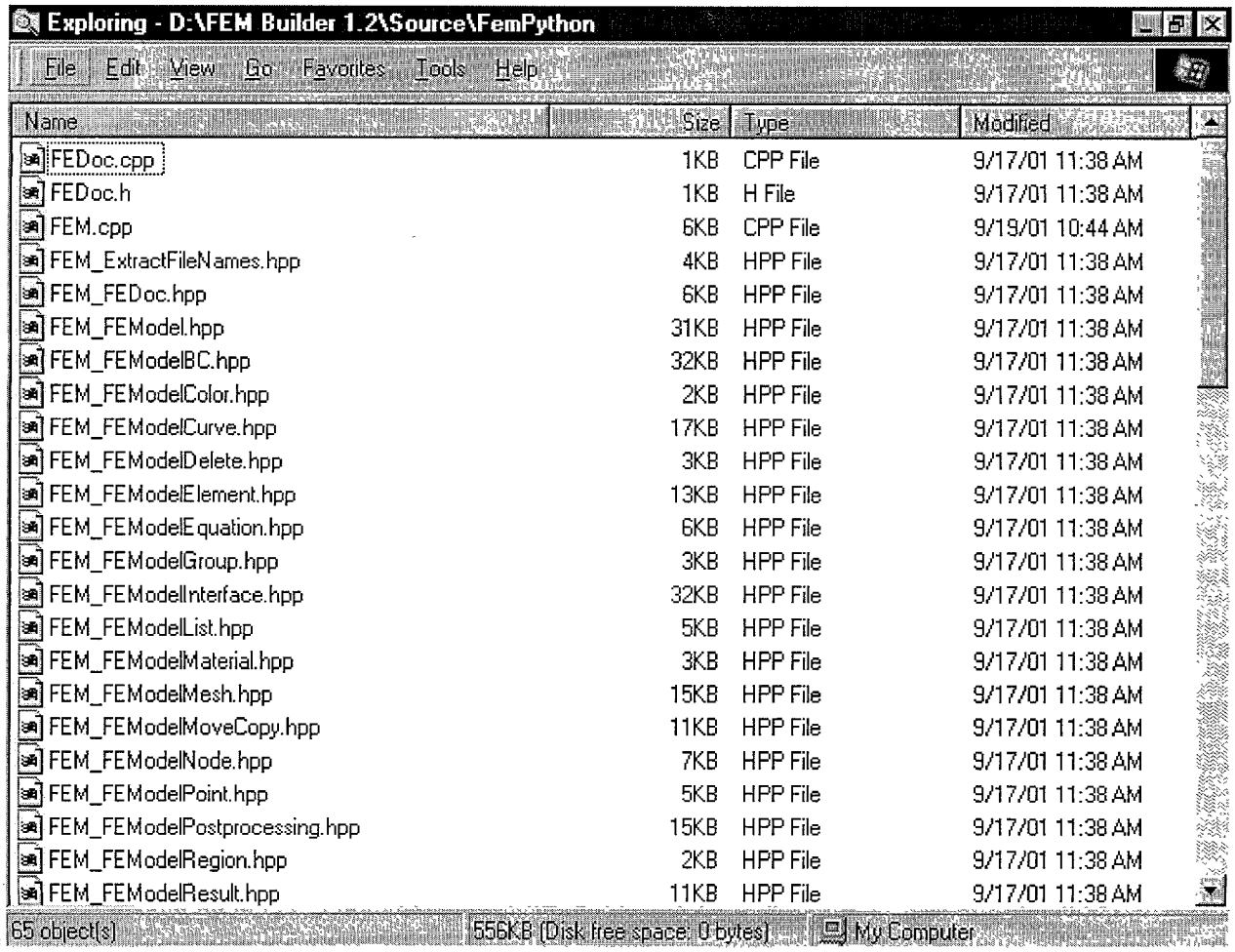

Exploring - D:\FEM Builder 1.2\Source\FemLib\Include
File Edit View Go Favorites Tools Help
Name Size Type Modified
CMoveCopy.h 5KB H File 9/17/01 11:37 AM
CNode.h 2KB H File 9/17/01 11:37 AM
COrient.h 2KB H File 9/17/01 11:37 AM
CQuadrant.h 5KB H File 9/17/01 11:37 AM
CRdCrackCombustionData.h 2KB H File 9/17/01 11:37 AM
CRefineMesh.h 7KB H File 9/17/01 11:37 AM
CResult.h 11KB H File 9/17/01 11:37 AM
CSindaAttrib.h 2KB H File 9/17/01 11:37 AM
CSindaNetwork.h 7KB H File 9/17/01 11:37 AM
CSubModel.h 2KB H File 9/17/01 11:37 AM
CSweepModel.h 3KB H File 9/17/01 11:37 AM
CThermalAblation.h 3KB H File 9/17/01 11:37 AM
CWrCrackCombustionData.h 2KB H File 9/17/01 11:37 AM
CWrSindaData.h 3KB H File 9/17/01 11:37 AM
FemEntityType.h 6KB H File 9/17/01 11:37 AM
FemStdIn.h 2KB H File 9/17/01 11:37 AM
GElementContour.h 3KB H File 9/17/01 11:37 AM
GElementDistortion.h 2KB H File 9/17/01 11:37 AM
GElementFun.h 3KB H File 9/17/01 11:37 AM
GElementGInt.h 4KB H File 9/17/01 11:37 AM
GElementInfo.h 3KB H File 9/17/01 11:37 AM
GElementNCIP.h 3KB H File 9/17/01 11:37 AM
GElementNCPP.h 2KB H File 9/17/01 11:38 AM
113 object(s)
351KB (Disk free space: 0 bytes)
My Computer

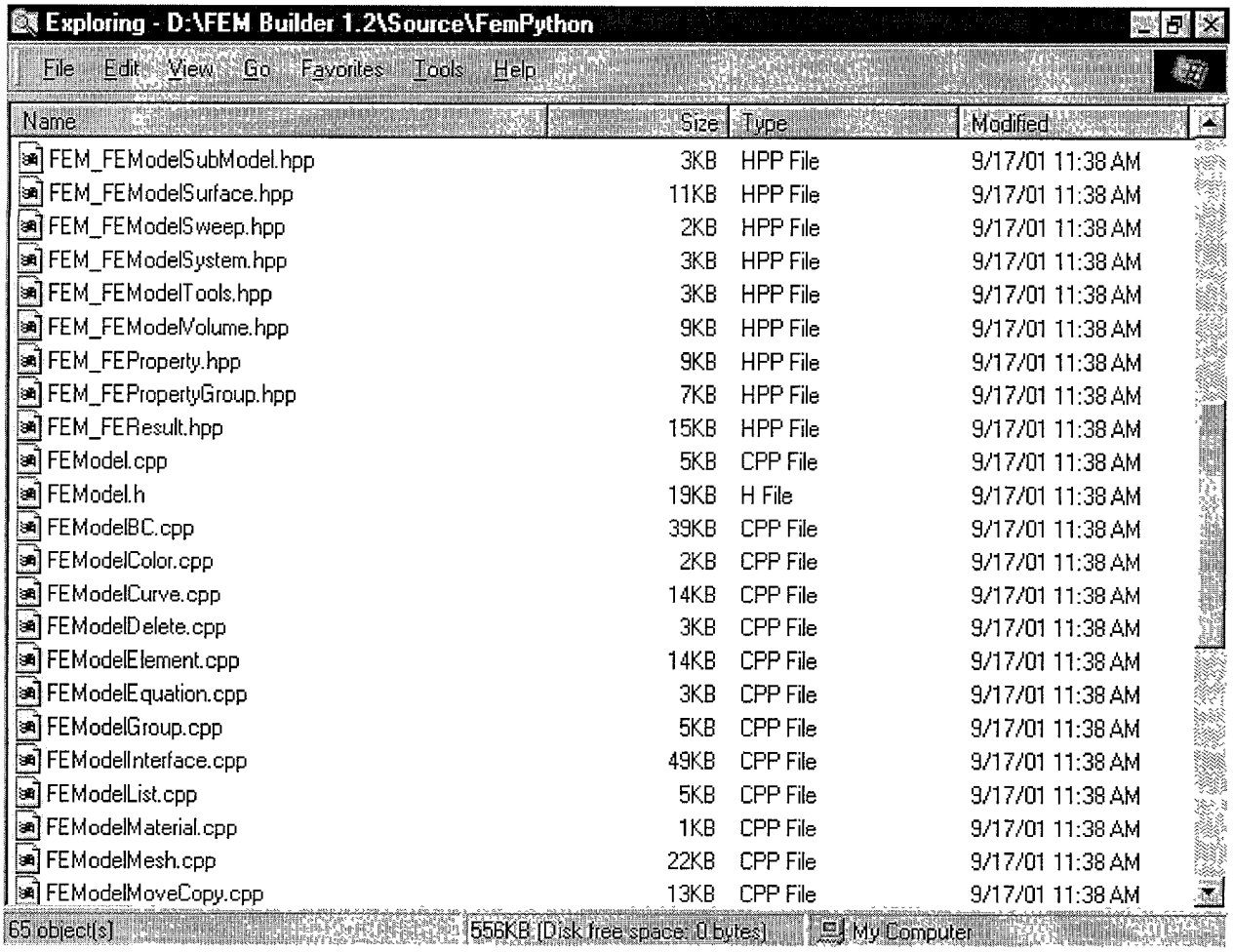

Exploring - D:\FEM Builder 1.2\Source\FemLib\Include
File Edit View Go Favorites Tools Help
Name Size Type Modified
GElementNCVI.h 2KB H File 9/17/01 11:38 AM
GElementOrient.h 1KB H File 9/17/01 11:38 AM
GElementSF.h 2KB H File 9/17/01 11:38 AM
GElementSFD.h 3KB H File 9/17/01 11:38 AM
GElementSFDD.h 4KB H File 9/17/01 11:38 AM
GMeshFun.h 1KB H File 9/17/01 11:38 AM
GPointFun.h 1KB H File 9/17/01 11:38 AM
GRdAnsysRes.h 3KB H File 9/17/01 11:38 AM
GSindaFun.h 1KB H File 9/17/01 11:38 AM
RdAbaqus.h 6KB H File 9/17/01 11:38 AM
RdAbaqusFIF.h 1KB H File 9/17/01 11:38 AM
RdAbaqusFOF.h 2KB H File 9/17/01 11:38 AM
RdAbaqusInput.h 3KB H File 9/17/01 11:38 AM
RdAnsysCDB.h 6KB H File 9/17/01 11:38 AM
RdAnsysRes.h 4KB H File 9/17/01 11:38 AM
RdAschar.h 5KB H File 9/17/01 11:38 AM
RdCCM.h 2KB H File 9/17/01 11:38 AM
RdCDCA.h 2KB H File 9/17/01 11:38 AM
RdFemDB.h 2KB H File 9/17/01 11:38 AM
RdFluent.h 5KB H File 9/17/01 11:38 AM
RdFluentJIO.h 3KB H File 9/17/01 11:38 AM
RdGridGen.h 2KB H File 9/17/01 11:38 AM
Start
Exploring - D:\FEM B...
untitled - Paint
5:08 PM

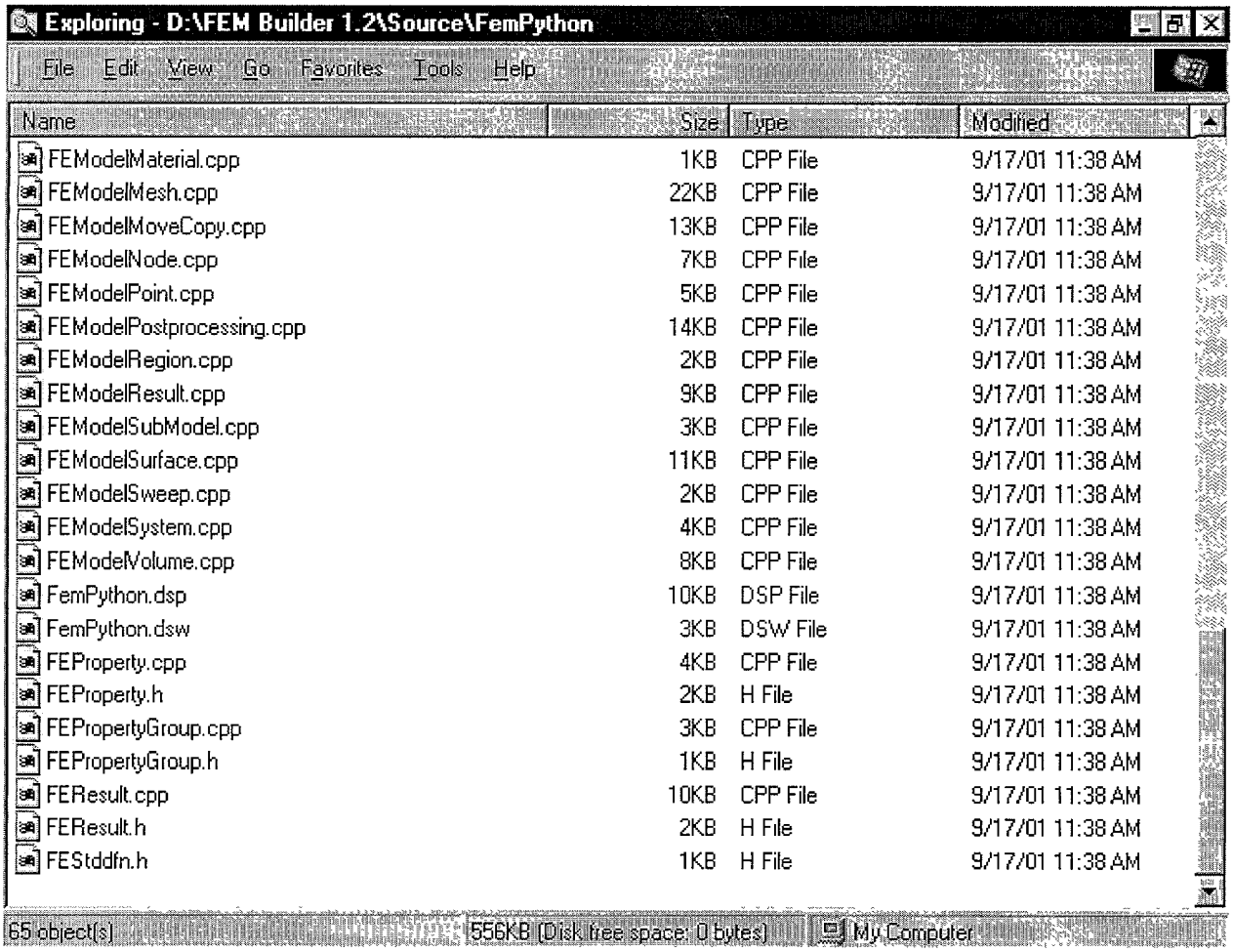

Exploring - D:\FEM Builder 1.2\Source\FemLib\Include
File Edit View Go Favorites Tools Help
Name Size Type Modified
RdIdeasMS.h 4KB H File 9/17/01 11:38 AM
RdPatran.h 3KB H File 9/17/01 11:38 AM
RdRecess.h 3KB H File 9/17/01 11:38 AM
RdSharp.h 3KB H File 9/17/01 11:38 AM
RdSinda.h 1KB H File 9/17/01 11:38 AM
RdSinGen.h 1KB H File 9/17/01 11:38 AM
RdTexChem.h 2KB H File 9/17/01 11:38 AM
RWAbaqus.h 1KB H File 9/17/01 11:38 AM
RWAnsys.h 2KB H File 9/17/01 11:38 AM
RWIdeasMS.h 2KB H File 9/17/01 11:38 AM
WrAbaqus.h 6KB H File 9/17/01 11:38 AM
WrAnsysCDB.h 3KB H File 9/17/01 11:38 AM
WrCCM.h 2KB H File 9/17/01 11:38 AM
WrCDCA.h 3KB H File 9/17/01 11:38 AM
WrFemDB.h 2KB H File 9/17/01 11:38 AM
WrFluent.h 8KB H File 9/17/01 11:38 AM
WrIdeasMS.h 4KB H File 9/17/01 11:38 AM
WrPatran.h 3KB H File 9/17/01 11:38 AM
WrPatranRes.h 2KB H File 9/17/01 11:38 AM
WrRecess.h 5KB H File 9/17/01 11:38 AM
WrSinda.h 2KB H File 9/17/01 11:38 AM
WrTexChem.h 3KB H File 9/17/01 11:38 AM
113 object(s) 351KB (Disk free space: 0 bytes) My Computer

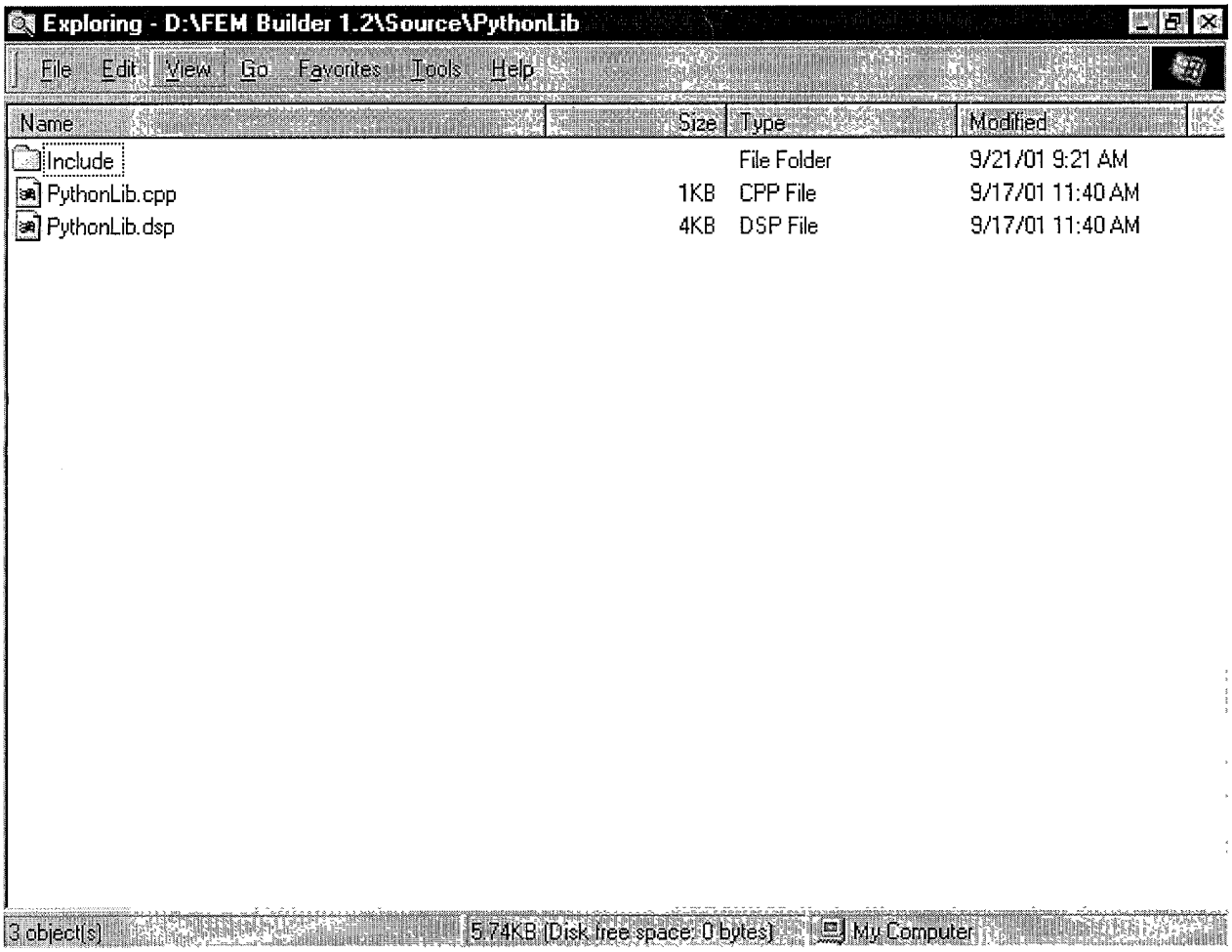

Exploring - D:\FEM Builder 1.2\Source\FemLib\Interfaces
File Edit View Go Favorites Tools Help
Name Size Type Modified
CRdCrackCombustionData.cpp 4KB CPP File 9/17/01 11:38 AM
CWrCrackCombustionData.cpp 2KB CPP File 9/17/01 11:38 AM
RdAbaqus.cpp 38KB CPP File 9/17/01 11:38 AM
RdAbaqusFIF.cpp 2KB CPP File 9/17/01 11:38 AM
RdAbaqusFOF.cpp 6KB CPP File 9/17/01 11:38 AM
RdAbaqusInput.cpp 45KB CPP File 9/17/01 11:38 AM
RdAnsysCDB.cpp 64KB CPP File 9/17/01 11:38 AM
RdAnsysRes.cpp 35KB CPP File 9/17/01 11:38 AM
RdAschar.cpp 21KB CPP File 9/17/01 11:38 AM
RdCCM.cpp 4KB CPP File 9/17/01 11:38 AM
RdCDCA.cpp 4KB CPP File 9/17/01 11:38 AM
RdFemDB.cpp 43KB CPP File 9/17/01 11:38 AM
RdFluent.cpp 46KB CPP File 9/17/01 11:38 AM
RdFluentJIO.cpp 23KB CPP File 9/17/01 11:38 AM
RdGridGen.cpp 6KB CPP File 9/17/01 11:38 AM
RdIdeasMS.cpp 49KB CPP File 9/17/01 11:38 AM
RdIdeasMSMaterial.cpp 27KB CPP File 9/17/01 11:38 AM
RdPatran.cpp 29KB CPP File 9/17/01 11:38 AM
RdRecess.cpp 37KB CPP File 9/17/01 11:38 AM
RdSharp.cpp 27KB CPP File 9/17/01 11:38 AM
RdSinda.cpp 5KB CPP File 9/17/01 11:38 AM
RdSinGen.cpp 4KB CPP File 9/17/01 11:38 AM
RdTexChem.cpp 15KB CPP File 9/17/01 11:38 AM
38 object(s) 936KB (Disk free space: 0 bytes) My Computer

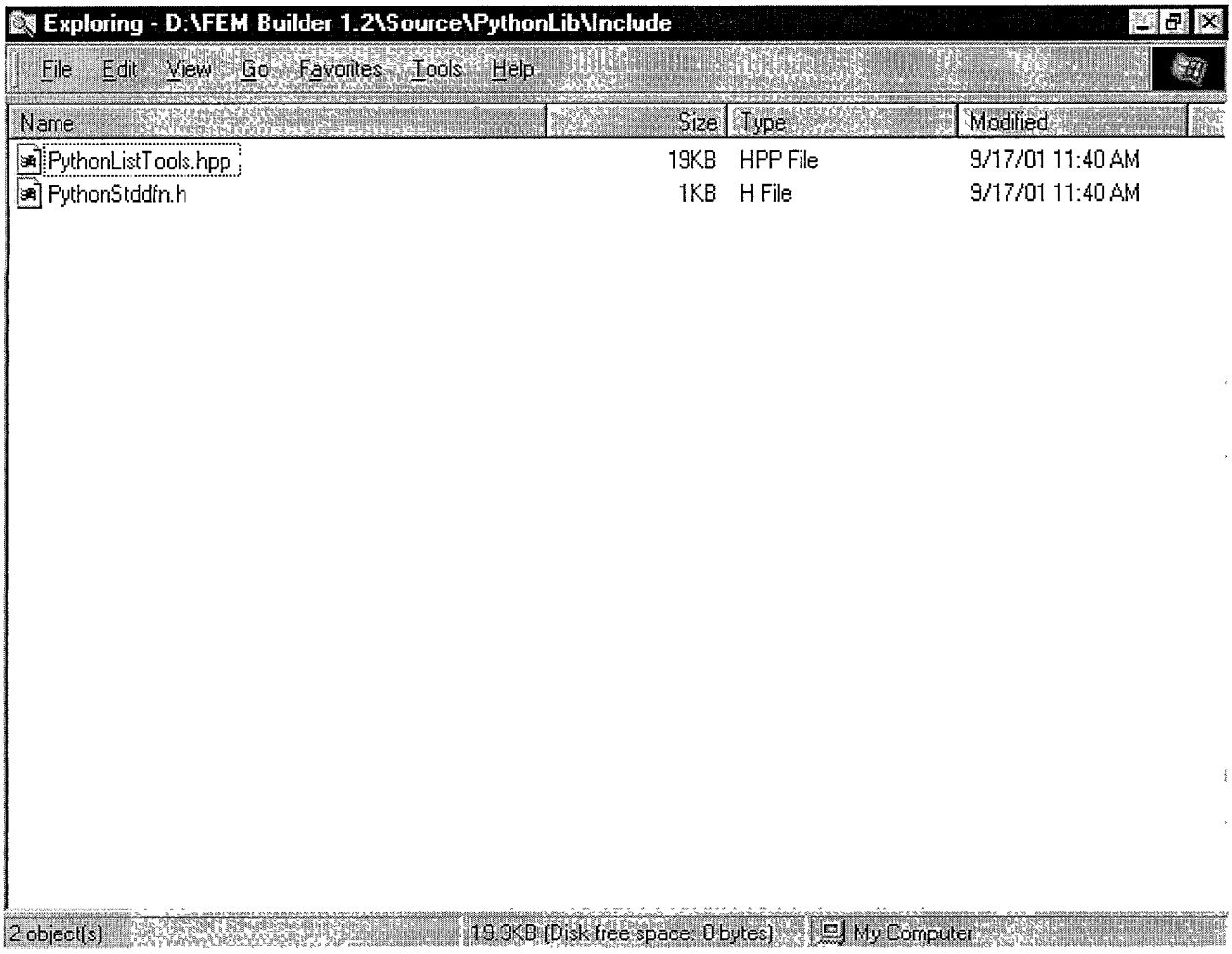

Exploring - D:\FEM Builder 1.2\Source\FemLib\Interfaces
File Edit View Go Favorites Tools Help
Name Size Type Modified
RdIdeasMSMaterial.cpp 27KB CPP File 9/17/01 11:38 AM
RdPatran.cpp 29KB CPP File 9/17/01 11:38 AM
RdRecess.cpp 37KB CPP File 9/17/01 11:38 AM
RdSharp.cpp 27KB CPP File 9/17/01 11:38 AM
RdSinda.cpp 5KB CPP File 9/17/01 11:38 AM
RdSinGen.cpp 4KB CPP File 9/17/01 11:38 AM
RdTexChem.cpp 15KB CPP File 9/17/01 11:38 AM
RWAbaqus.cpp 10KB CPP File 9/17/01 11:38 AM
RWAnsys.cpp 20KB CPP File 9/17/01 11:38 AM
RWIdeasMS.cpp 19KB CPP File 9/17/01 11:38 AM
WrAbaqus.cpp 36KB CPP File 9/17/01 11:38 AM
WrAnsysCDB.cpp 34KB CPP File 9/17/01 11:38 AM
WrCCM.cpp 6KB CPP File 9/17/01 11:38 AM
WrCDCA.cpp 12KB CPP File 9/17/01 11:38 AM
WrFemDB.cpp 37KB CPP File 9/17/01 11:38 AM
WrFluent.cpp 65KB CPP File 9/17/01 11:38 AM
WrIdeasMS.cpp 44KB CPP File 9/17/01 11:38 AM
WrIdeasMSMaterial.cpp 27KB CPP File 9/17/01 11:38 AM
WrPatran.cpp 33KB CPP File 9/17/01 11:38 AM
WrRecess.cpp 35KB CPP File 9/17/01 11:38 AM
WrSinda.cpp 19KB CPP File 9/17/01 11:38 AM
WrTexChem.cpp 22KB CPP File 9/17/01 11:38 AM
38 object(s) 938KB (Disk free space: 0 bytes) My Computer

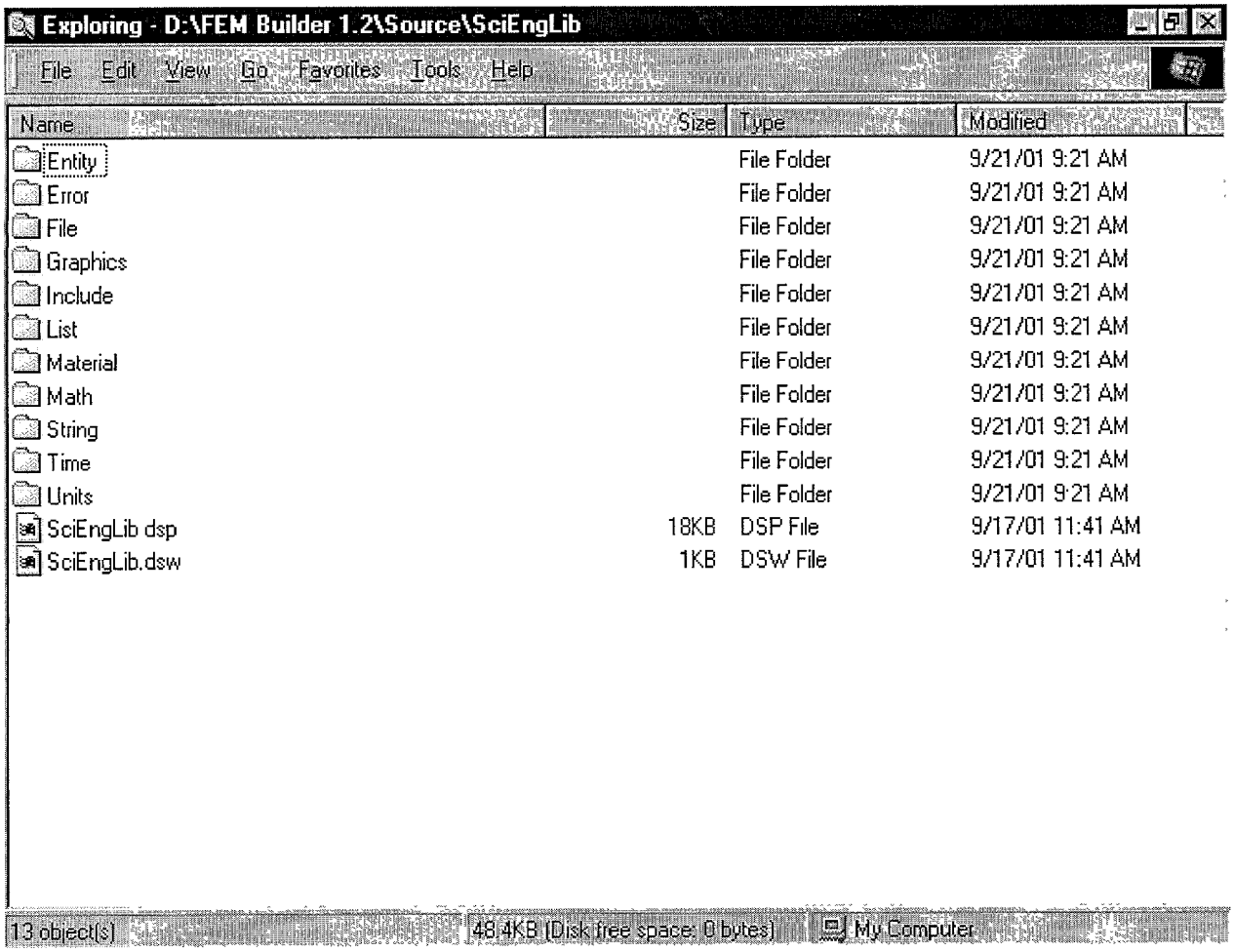

Exploring - D:\FEM Builder 1.2\Source\FemPython
File Edit View Go Favorites Tools Help
Name Size Type Modified
FEDoc.cpp 1KB CPP File 9/17/01 11:38 AM
FEDoc.h 1KB H File 9/17/01 11:38 AM
FEM.cpp 6KB CPP File 9/19/01 10:44 AM
FEM_ExtractFileNames.hpp 4KB HPP File 9/17/01 11:38 AM
FEM_FEDoc.hpp 6KB HPP File 9/17/01 11:38 AM
FEM_FEModel.hpp 31KB HPP File 9/17/01 11:38 AM
FEM_FEModelBC.hpp 32KB HPP File 9/17/01 11:38 AM
FEM_FEModelColor.hpp 2KB HPP File 9/17/01 11:38 AM
FEM_FEModelCurve.hpp 17KB HPP File 9/17/01 11:38 AM
FEM_FEModelDelete.hpp 3KB HPP File 9/17/01 11:38 AM
FEM_FEModelElement.hpp 13KB HPP File 9/17/01 11:38 AM
FEM_FEModelEquation.hpp 6KB HPP File 9/17/01 11:38 AM
FEM_FEModelGroup.hpp 3KB HPP File 9/17/01 11:38 AM
FEM_FEModelInterface.hpp 32KB HPP File 9/17/01 11:38 AM
FEM_FEModelList.hpp 5KB HPP File 9/17/01 11:38 AM
FEM_FEModelMaterial.hpp 3KB HPP File 9/17/01 11:38 AM
FEM_FEModelMesh.hpp 15KB HPP File 9/17/01 11:38 AM
FEM_FEModelMoveCopy.hpp 11KB HPP File 9/17/01 11:38 AM
FEM_FEModelNode.hpp 7KB HPP File 9/17/01 11:38 AM
FEM_FEModelPoint.hpp 5KB HPP File 9/17/01 11:38 AM
FEM_FEModelPostprocessing.hpp 15KB HPP File 9/17/01 11:38 AM
FEM_FEModelRegion.hpp 2KB HPP File 9/17/01 11:38 AM
FEM_FEModelResult.hpp 11KB HPP File 9/17/01 11:38 AM
65 object(s) 556KB (Disk free space: 0 bytes) My Computer

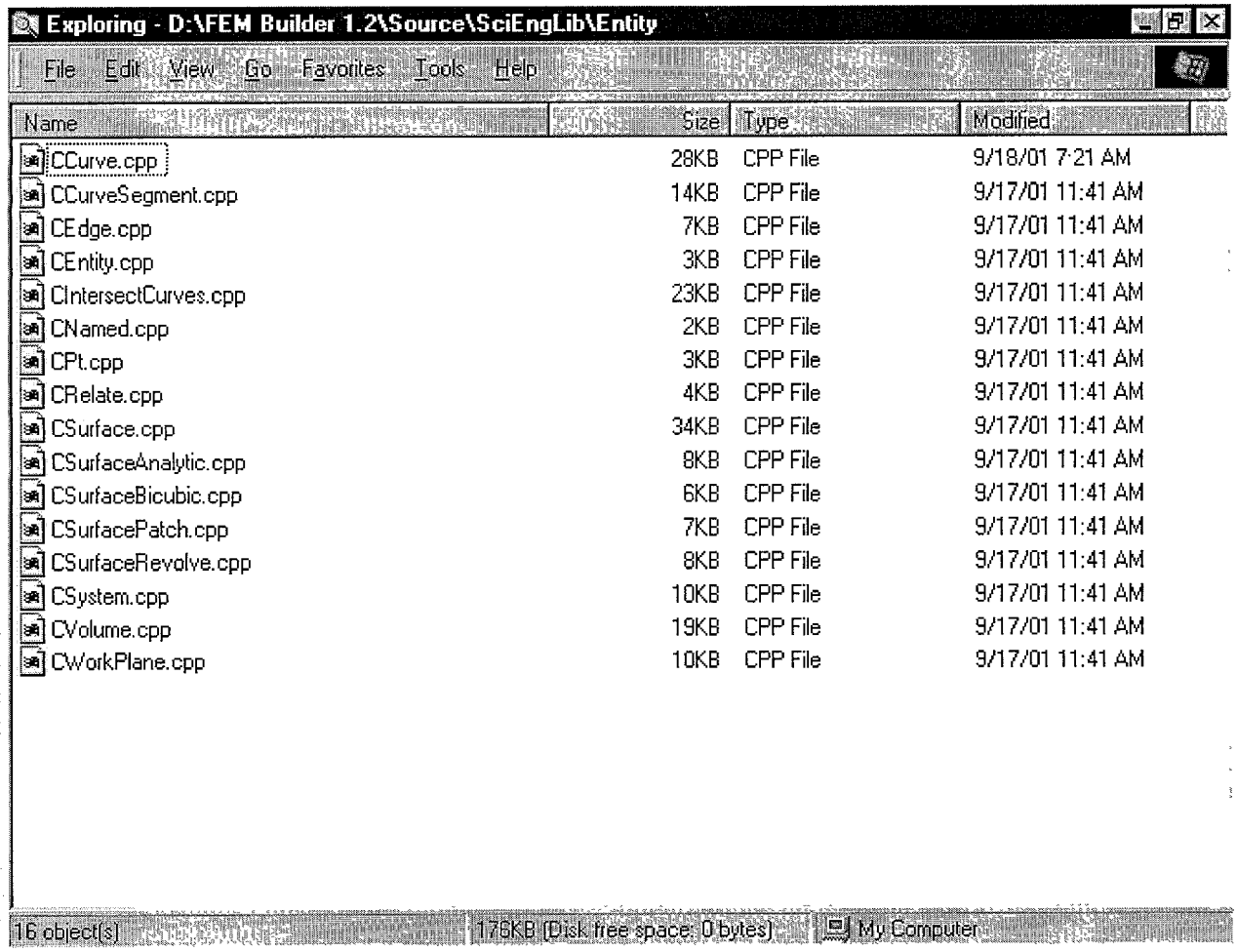

Exploring - D:\FEM Builder 1.2\Source\FemPython
File Edit View Go Favorites Tools Help
Name Size Type Modified
FEM_FEModelSubModel.hpp 3KB HPP File 9/17/01 11:38 AM
FEM_FEModelSurface.hpp 11KB HPP File 9/17/01 11:38 AM
FEM_FEModelSweep.hpp 2KB HPP File 9/17/01 11:38 AM
FEM_FEModelSystem.hpp 3KB HPP File 9/17/01 11:38 AM
FEM_FEModelTools.hpp 3KB HPP File 9/17/01 11:38 AM
FEM_FEModelVolume.hpp 9KB HPP File 9/17/01 11:38 AM
FEM_FEProperty.hpp 9KB HPP File 9/17/01 11:38 AM
FEM_FEPropertyGroup.hpp 7KB HPP File 9/17/01 11:38 AM
FEM_FEResult.hpp 15KB HPP File 9/17/01 11:38 AM
FEModel.cpp 5KB CPP File 9/17/01 11:38 AM
FEModel.h 19KB H File 9/17/01 11:38 AM
FEModelBC.cpp 39KB CPP File 9/17/01 11:38 AM
FEModelColor.cpp 2KB CPP File 9/17/01 11:38 AM
FEModelCurve.cpp 14KB CPP File 9/17/01 11:38 AM
FEModelDelete.cpp 3KB CPP File 9/17/01 11:38 AM
FEModelElement.cpp 14KB CPP File 9/17/01 11:38 AM
FEModelEquation.cpp 3KB CPP File 9/17/01 11:38 AM
FEModelGroup.cpp 5KB CPP File 9/17/01 11:38 AM
FEModelInterface.cpp 49KB CPP File 9/17/01 11:38 AM
FEModelList.cpp 5KB CPP File 9/17/01 11:38 AM
FEModelMaterial.cpp 1KB CPP File 9/17/01 11:38 AM
FEModelMesh.cpp 22KB CPP File 9/17/01 11:38 AM
FEModelMoveCopy.cpp 13KB CPP File 9/17/01 11:38 AM
65 object(s) 556KB (Disk free space: 0 bytes) My Computer

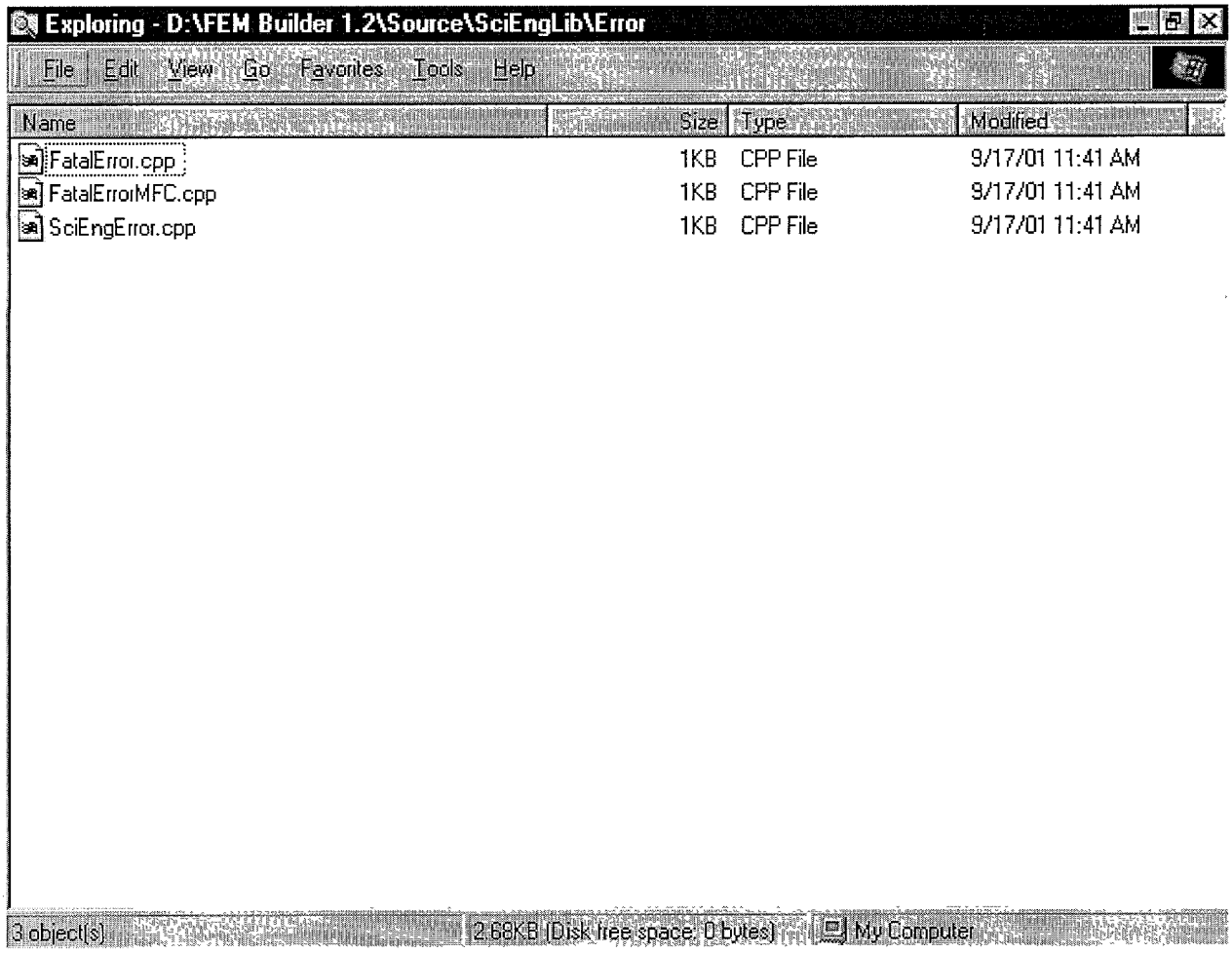

Exploring - D:\FEM Builder 1.2\Source\FemPython
File Edit View Go Favorites Tools Help
Name Size Type Modified
FEModelMaterial.cpp 1KB CPP File 9/17/01 11:38 AM
FEModelMesh.cpp 22KB CPP File 9/17/01 11:38 AM
FEModelMoveCopy.cpp 13KB CPP File 9/17/01 11:38 AM
FEModelNode.cpp 7KB CPP File 9/17/01 11:38 AM
FEModelPoint.cpp 5KB CPP File 9/17/01 11:38 AM
FEModelPostprocessing.cpp 14KB CPP File 9/17/01 11:38 AM
FEModelRegion.cpp 2KB CPP File 9/17/01 11:38 AM
FEModelResult.cpp 9KB CPP File 9/17/01 11:38 AM
FEModelSubModel.cpp 3KB CPP File 9/17/01 11:38 AM
FEModelSurface.cpp 11KB CPP File 9/17/01 11:38 AM
FEModelSweep.cpp 2KB CPP File 9/17/01 11:38 AM
FEModelSystem.cpp 4KB CPP File 9/17/01 11:38 AM
FEModelVolume.cpp 8KB CPP File 9/17/01 11:38 AM
FemPython.dsp 10KB DSP File 9/17/01 11:38 AM
FemPython.dsw 3KB DSW File 9/17/01 11:38 AM
FEProperty.cpp 4KB CPP File 9/17/01 11:38 AM
FEProperty.h 2KB H File 9/17/01 11:38 AM
FEPropertyGroup.cpp 3KB CPP File 9/17/01 11:38 AM
FEPropertyGroup.h 1KB H File 9/17/01 11:38 AM
FEResult.cpp 10KB CPP File 9/17/01 11:38 AM
FEResult.h 2KB H File 9/17/01 11:38 AM
FEStddfn.h 1KB H File 9/17/01 11:38 AM
65 object(s) 556KB (Disk free space: 0 bytes) My Computer

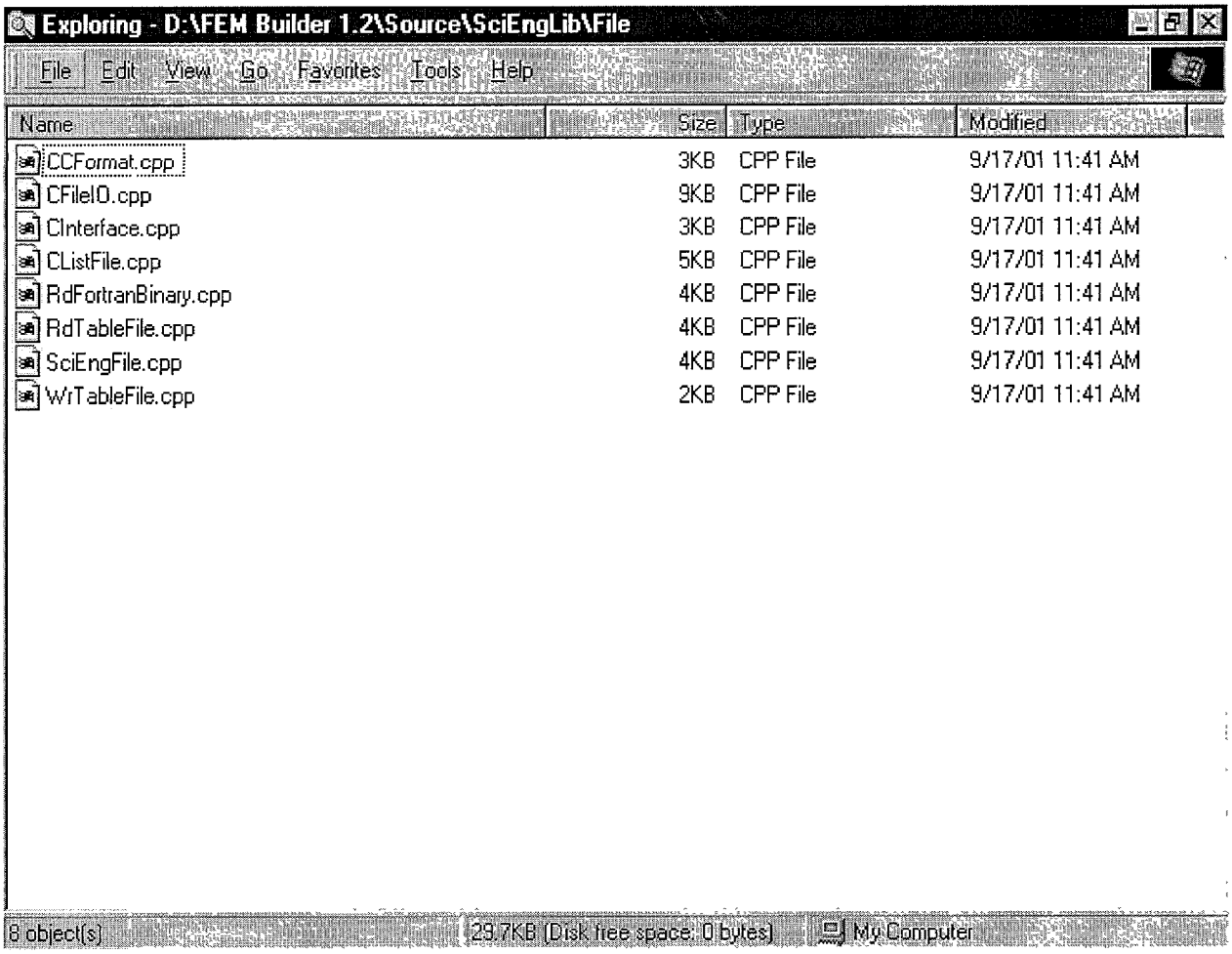

Exploring - D:\FEM Builder 1.2\Source\PythonLib
File Edit View Go Favorites Tools Help
Name Size Type Modified
Include File Folder 9/21/01 9:21 AM
PythonLib.cpp 1KB CPP File 9/17/01 11:40 AM
PythonLib.dsp 4KB DSP File 9/17/01 11:40 AM
3 object(s) 5.74KB (Disk free space: 0 bytes) My Computer

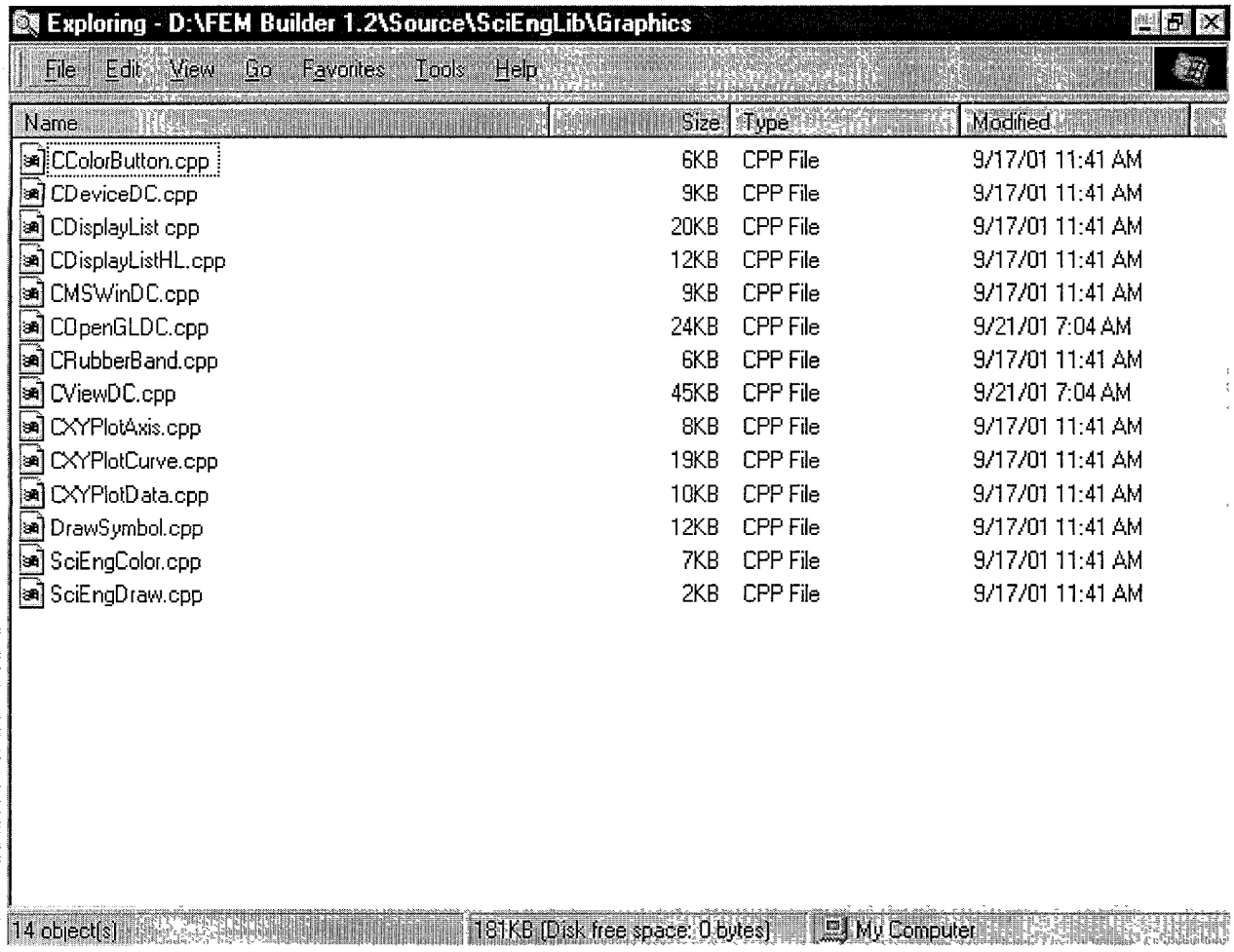

Exploring - D:\FEM Builder 1.2\Source\PythonLib\Include
File Edit View Go Favorites Tools Help
Name Size Type Modified
PythonListTools.hpp 19KB HPP File 9/17/01 11:40 AM
PythonStddfn.h 1KB H File 9/17/01 11:40 AM
2 object(s) 19.3KB (Disk free space: 0 bytes) My Computer

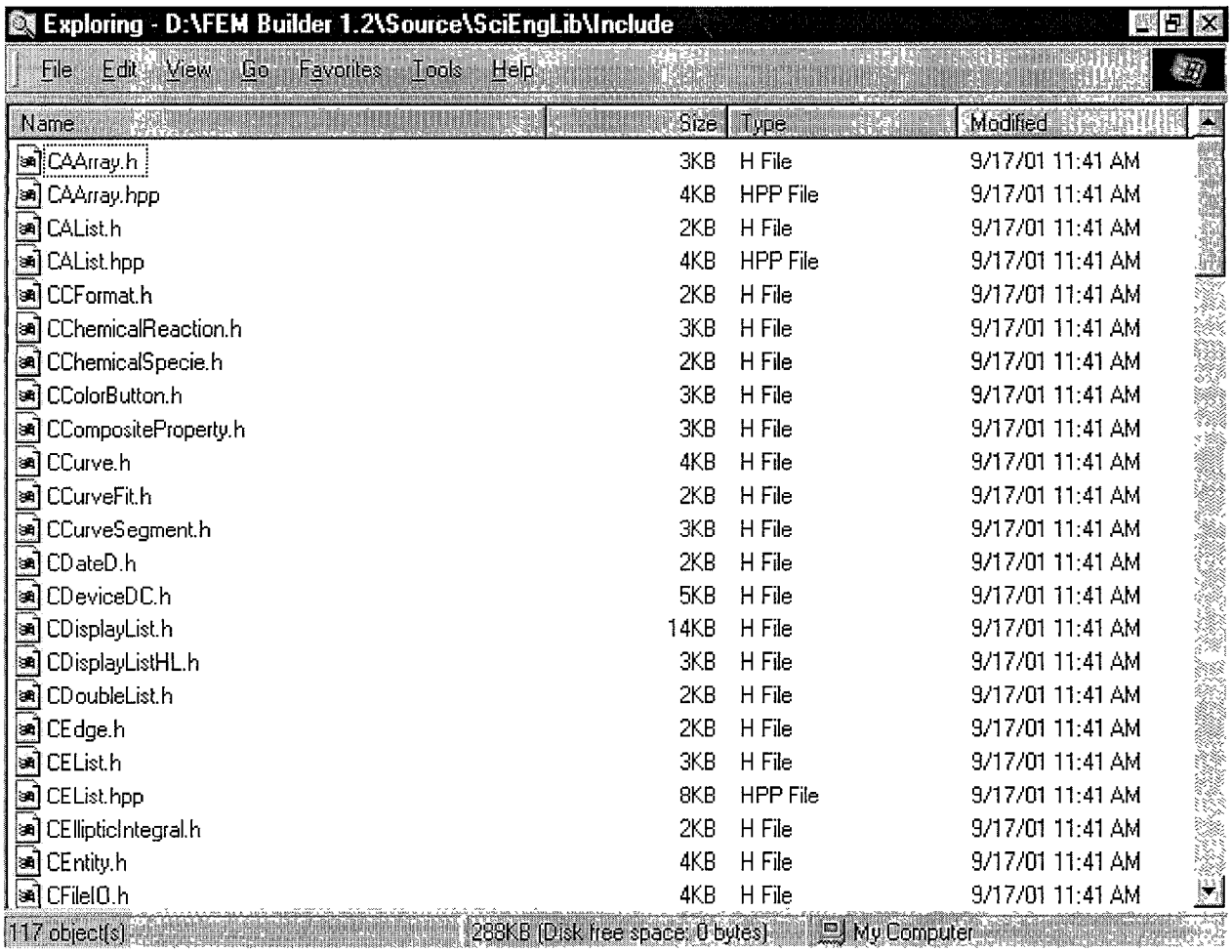

Exploring - D:\FEM Builder 1.2\Source\SciEngLib
File Edit View Go Favorites Tools Help
Name Size Type Modified
Entity File Folder 9/21/01 9:21 AM
Error File Folder 9/21/01 9:21 AM
File File Folder 9/21/01 9:21 AM
Graphics File Folder 9/21/01 9:21 AM
Include File Folder 9/21/01 9:21 AM
List File Folder 9/21/01 9:21 AM
Material File Folder 9/21/01 9:21 AM
Math File Folder 9/21/01 9:21 AM
String File Folder 9/21/01 9:21 AM
Time File Folder 9/21/01 9:21 AM
Units File Folder 9/21/01 9:21 AM
SciEngLib dsp 18KB DSP File 9/17/01 11:41 AM
SciEngLib.dsw 1KB DSW File 9/17/01 11:41 AM
13 object(s)
48.4KB (Disk free space: 0 bytes)
My Computer

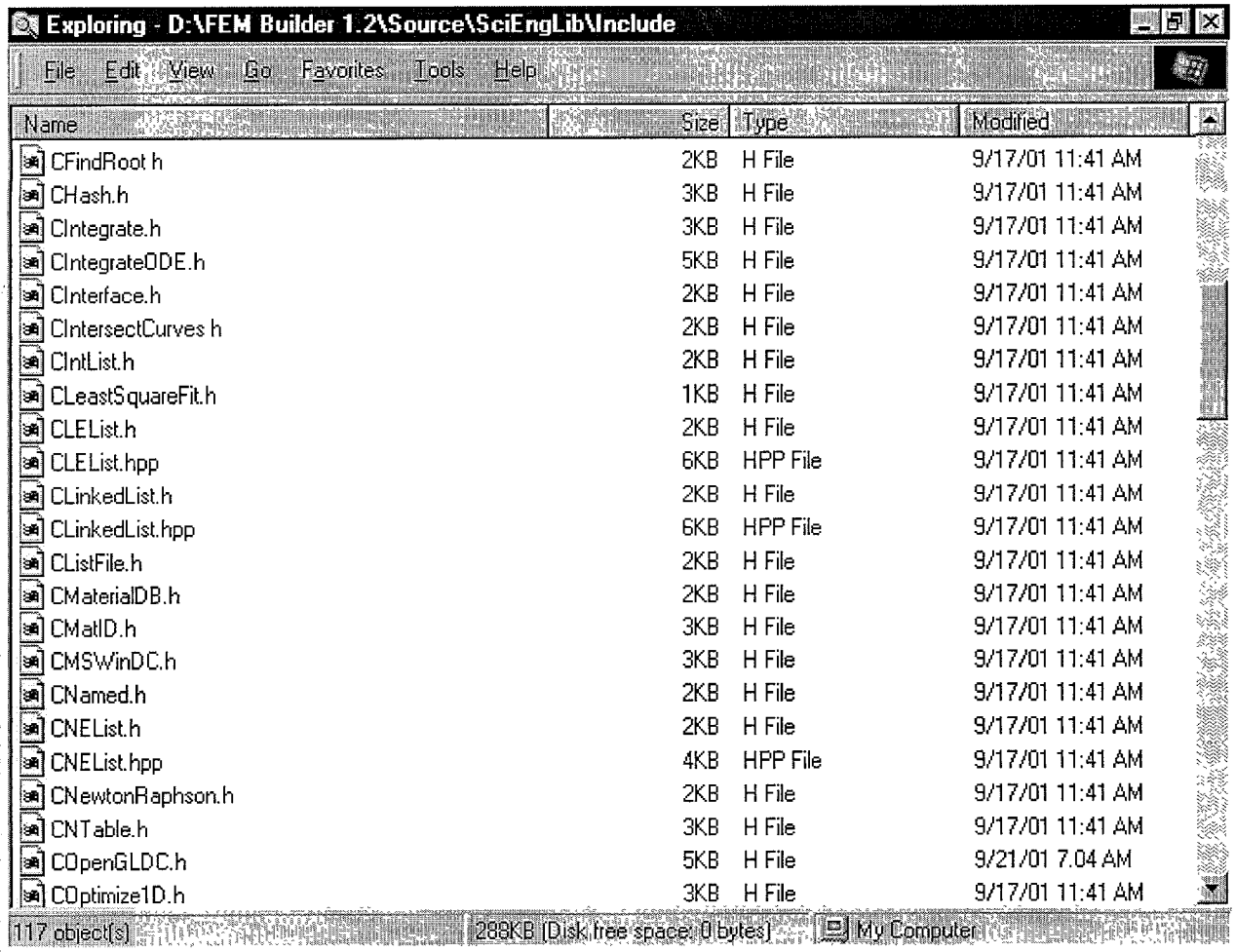

Exploring - D:\FEM Builder 1.2\Source\SciEngLib\Entity
File Edit View Go Favorites Tools Help
Name Size Type Modified
CCurve.cpp 28KB CPP File 9/18/01 7:21 AM
CCurveSegment.cpp 14KB CPP File 9/17/01 11:41 AM
CEdge.cpp 7KB CPP File 9/17/01 11:41 AM
CEntity.cpp 3KB CPP File 9/17/01 11:41 AM
CIntersectCurves.cpp 23KB CPP File 9/17/01 11:41 AM
CNamed.cpp 2KB CPP File 9/17/01 11:41 AM
CPt.cpp 3KB CPP File 9/17/01 11:41 AM
CRelate.cpp 4KB CPP File 9/17/01 11:41 AM
CSurface.cpp 34KB CPP File 9/17/01 11:41 AM
CSurfaceAnalytic.cpp 8KB CPP File 9/17/01 11:41 AM
CSurfaceBicubic.cpp 6KB CPP File 9/17/01 11:41 AM
CSurfacePatch.cpp 7KB CPP File 9/17/01 11:41 AM
CSurfaceRevolve.cpp 8KB CPP File 9/17/01 11:41 AM
CSystem.cpp 10KB CPP File 9/17/01 11:41 AM
CVolume.cpp 19KB CPP File 9/17/01 11:41 AM
CWorkPlane.cpp 10KB CPP File 9/17/01 11:41 AM
16 object(s) 178KB (Disk free space: 0 bytes) My Computer

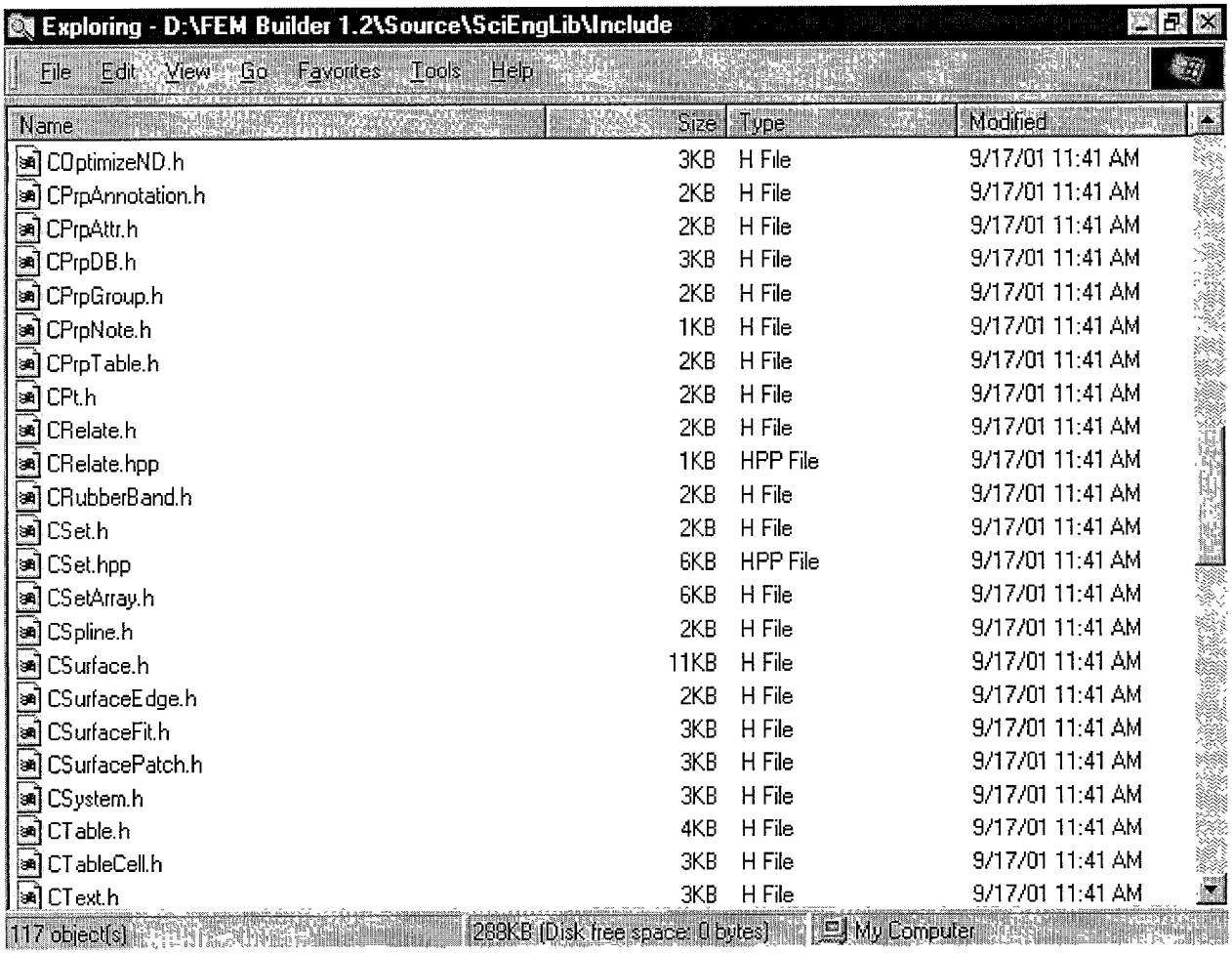

Exploring - D:\FEM Builder 1.2\Source\SciEngLib\Error
File Edit View Go Favorites Tools Help
Name Size Type Modified
FatalError.cpp 1KB CPP File 9/17/01 11:41 AM
FatalErrorMFC.cpp 1KB CPP File 9/17/01 11:41 AM
SciEngError.cpp 1KB CPP File 9/17/01 11:41 AM
3 object(s) 2.68KB (Disk free space: 0 bytes) My Computer

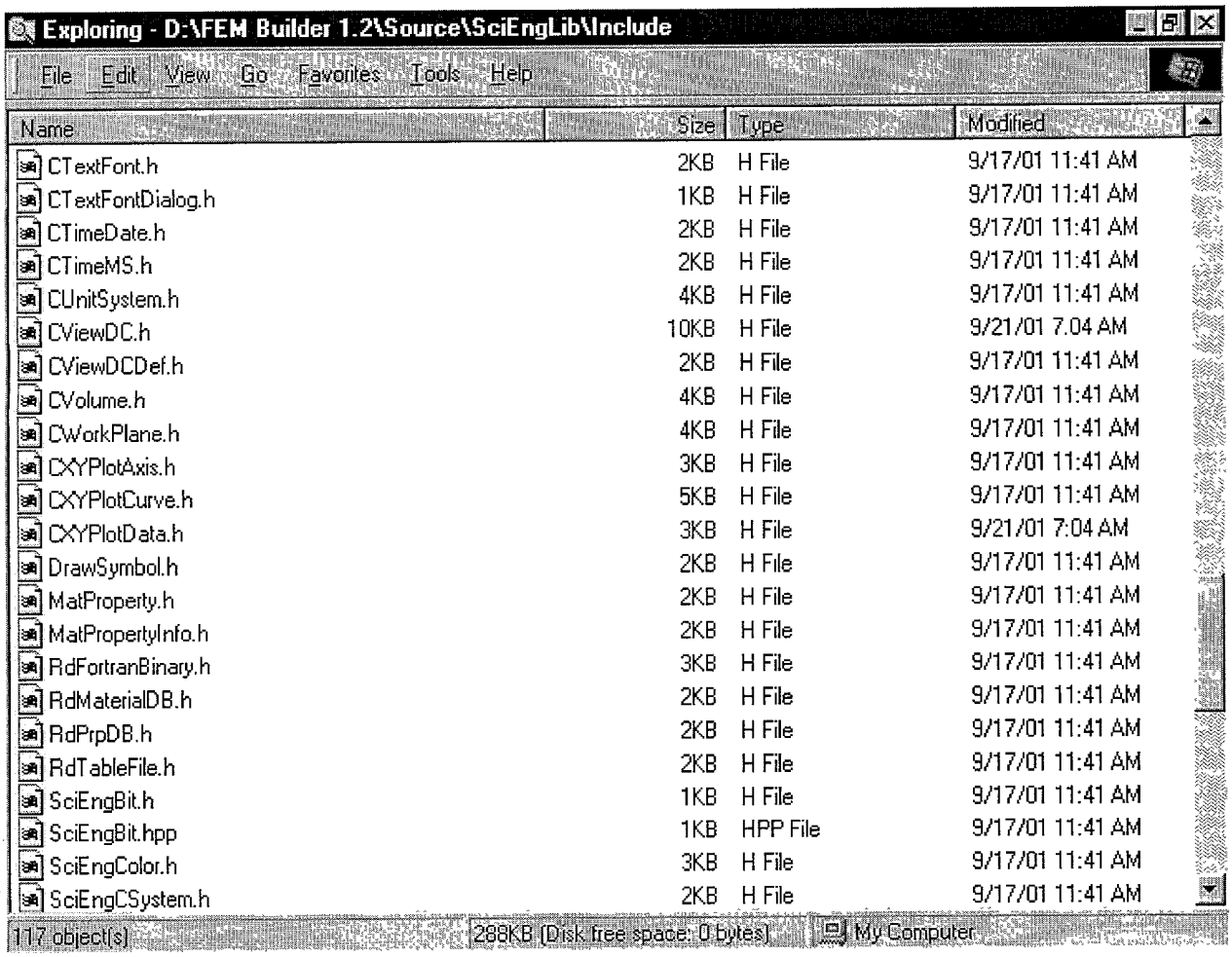

Exploring - D:\FEM Builder 1.2\Source\SciEngLib\File
File Edit View Go Favorites Tools Help
Name Size Type Modified
CCFormat.cpp 3KB CPP File 9/17/01 11:41 AM
CFileIO.cpp 9KB CPP File 9/17/01 11:41 AM
CInterface.cpp 3KB CPP File 9/17/01 11:41 AM
CListFile.cpp 5KB CPP File 9/17/01 11:41 AM
RdFortranBinary.cpp 4KB CPP File 9/17/01 11:41 AM
RdTableFile.cpp 4KB CPP File 9/17/01 11:41 AM
SciEngFile.cpp 4KB CPP File 9/17/01 11:41 AM
WrTableFile.cpp 2KB CPP File 9/17/01 11:41 AM
8 object(s)
29.7KB (Disk free space: 0 bytes)
My Computer

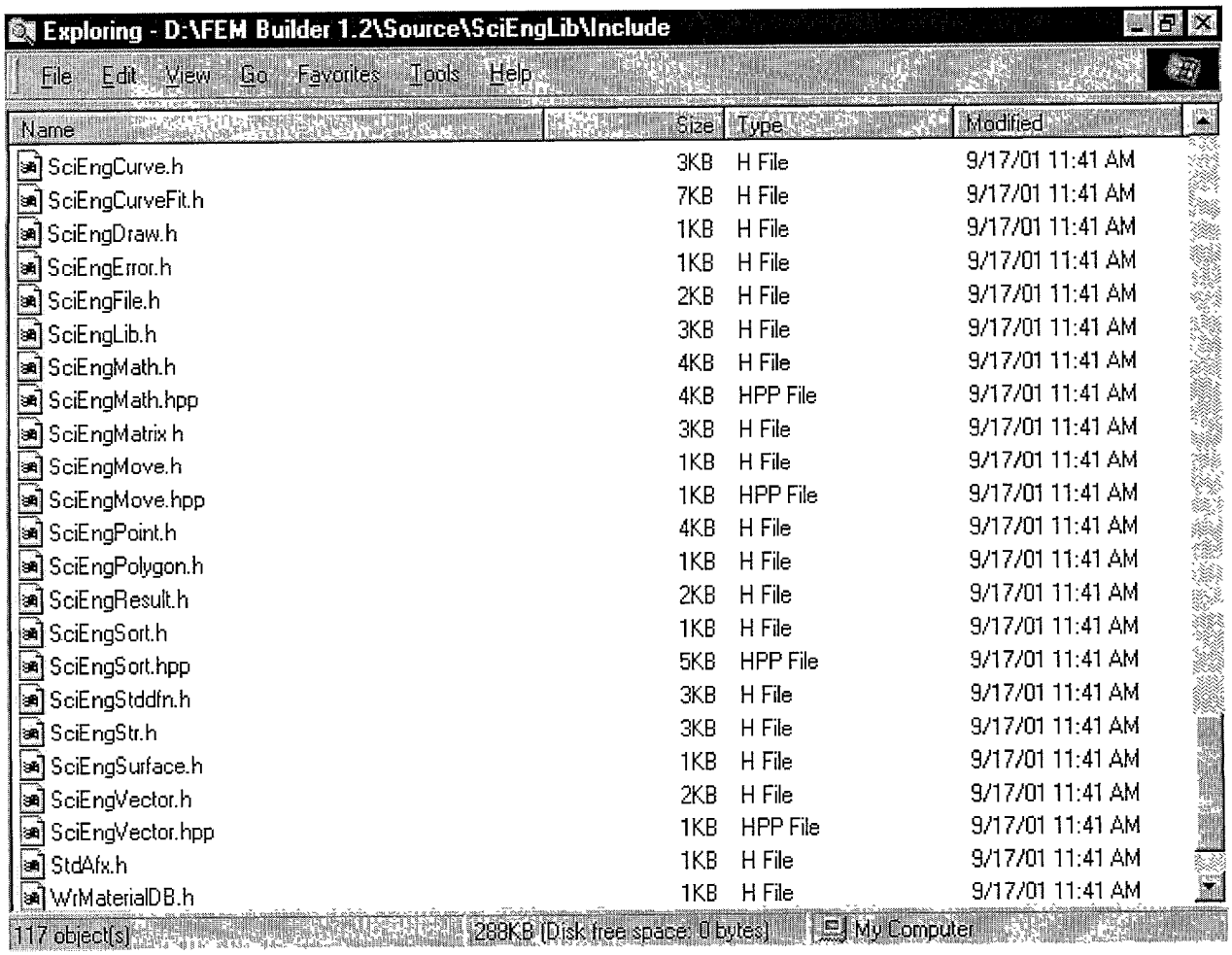

Exploring - D:\FEM Builder 1.2\Source\SciEngLib\Graphics
File Edit View Go Favorites Tools Help
Name Size Type Modified
CColorButton.cpp 6KB CPP File 9/17/01 11:41 AM
CDeviceDC.cpp 9KB CPP File 9/17/01 11:41 AM
CDisplayList.cpp 20KB CPP File 9/17/01 11:41 AM
CDisplayListHL.cpp 12KB CPP File 9/17/01 11:41 AM
CMSWinDC.cpp 9KB CPP File 9/17/01 11:41 AM
COpenGLDC.cpp 24KB CPP File 9/21/01 7:04 AM
CRubberBand.cpp 6KB CPP File 9/17/01 11:41 AM
CViewDC.cpp 45KB CPP File 9/21/01 7:04 AM
CXYPlotAxis.cpp 8KB CPP File 9/17/01 11:41 AM
CXYPlotCurve.cpp 19KB CPP File 9/17/01 11:41 AM
CXYPlotData.cpp 10KB CPP File 9/17/01 11:41 AM
DrawSymbol.cpp 12KB CPP File 9/17/01 11:41 AM
SciEngColor.cpp 7KB CPP File 9/17/01 11:41 AM
SciEngDraw.cpp 2KB CPP File 9/17/01 11:41 AM
14 object(s) 181KB (Disk free space: 0 bytes) My Computer

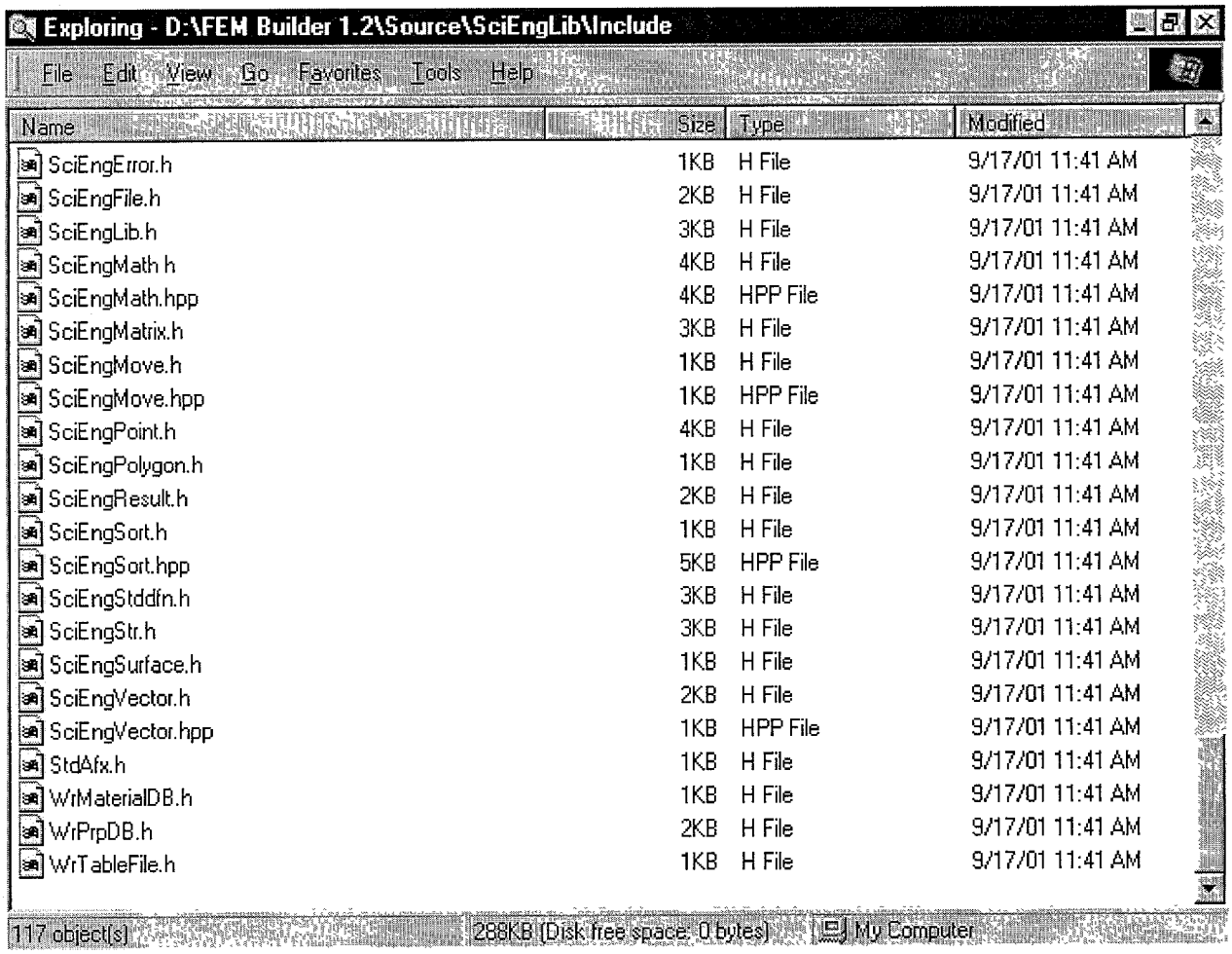

Exploring - D:\FEM Builder 1.2\Source\SciEngLib\Include
File Edit View Go Favorites Tools Help
Name Size Type Modified
CAArray.h 3KB H File 9/17/01 11:41 AM
CAArray.hpp 4KB HPP File 9/17/01 11:41 AM
CAList.h 2KB H File 9/17/01 11:41 AM
CAList.hpp 4KB HPP File 9/17/01 11:41 AM
CCFormat.h 2KB H File 9/17/01 11:41 AM
CChemicalReaction.h 3KB H File 9/17/01 11:41 AM
CChemicalSpecie.h 2KB H File 9/17/01 11:41 AM
CColorButton.h 3KB H File 9/17/01 11:41 AM
CCompositeProperty.h 3KB H File 9/17/01 11:41 AM
CCurve.h 4KB H File 9/17/01 11:41 AM
CCurveFit.h 2KB H File 9/17/01 11:41 AM
CCurveSegment.h 3KB H File 9/17/01 11:41 AM
CDateD.h 2KB H File 9/17/01 11:41 AM
CDeviceDC.h 5KB H File 9/17/01 11:41 AM
CDisplayList.h 14KB H File 9/17/01 11:41 AM
CDisplayListHL.h 3KB H File 9/17/01 11:41 AM
CDoubleList.h 2KB H File 9/17/01 11:41 AM
CEdge.h 2KB H File 9/17/01 11:41 AM
CEList.h 3KB H File 9/17/01 11:41 AM
CEList.hpp 8KB HPP File 9/17/01 11:41 AM
CEllipticIntegral.h 2KB H File 9/17/01 11:41 AM
CEntity.h 4KB H File 9/17/01 11:41 AM
CFileIO.h 4KB H File 9/17/01 11:41 AM
117 object(s) 288KB (Disk free space: 0 bytes) My Computer

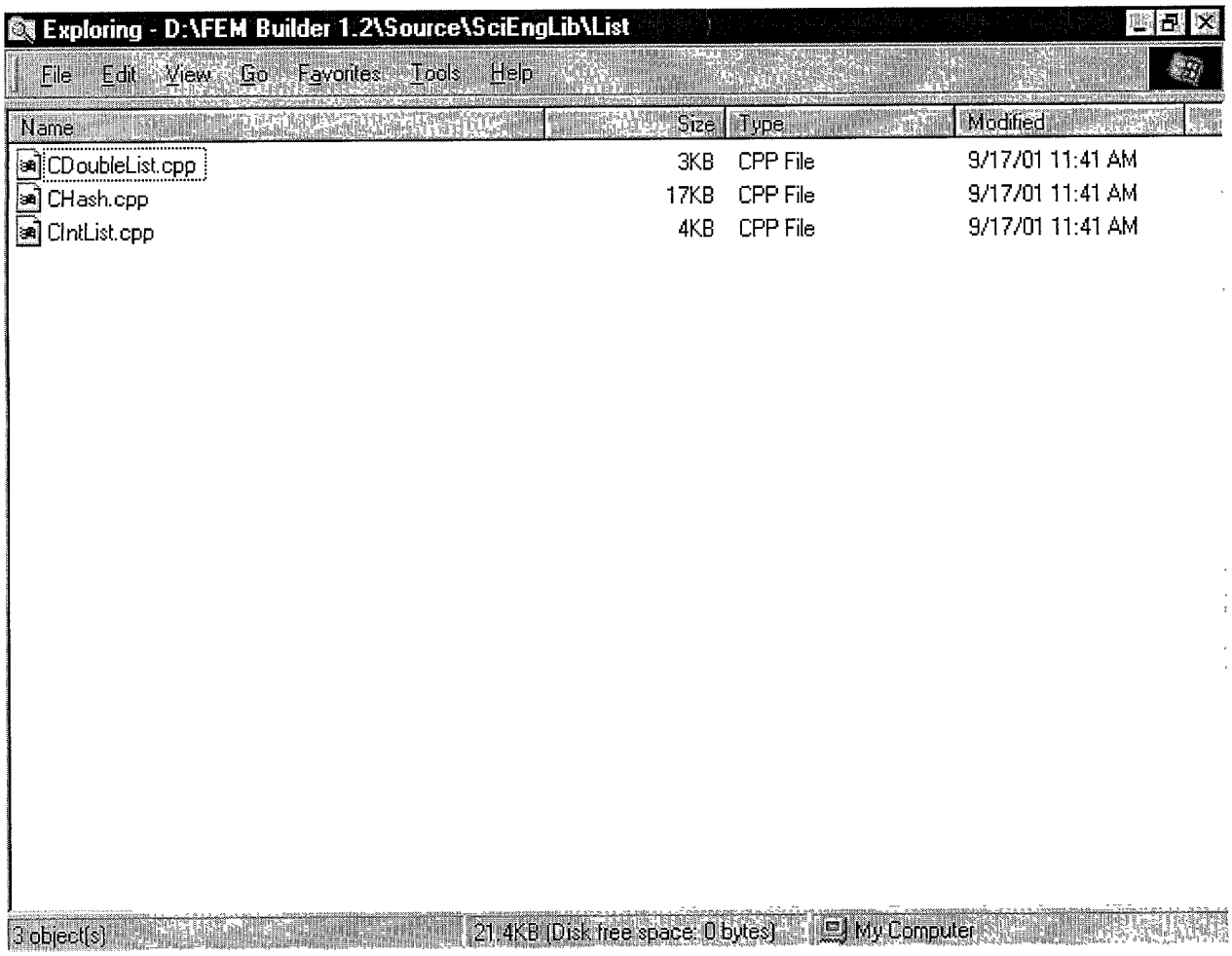

Exploring - D:\FEM Builder 1.2\Source\SciEngLib\Include
File Edit View Go Favorites Tools Help
Name Size Type Modified
CFindRoot.h 2KB H File 9/17/01 11:41 AM
CHash.h 3KB H File 9/17/01 11:41 AM
CIntegrate.h 3KB H File 9/17/01 11:41 AM
CIntegrateODE.h 5KB H File 9/17/01 11:41 AM
CInterface.h 2KB H File 9/17/01 11:41 AM
CIntersectCurves.h 2KB H File 9/17/01 11:41 AM
CIntList.h 2KB H File 9/17/01 11:41 AM
CLeastSquareFit.h 1KB H File 9/17/01 11:41 AM
CLEList.h 2KB H File 9/17/01 11:41 AM
CLEList.hpp 6KB HPP File 9/17/01 11:41 AM
CLinkedList.h 2KB H File 9/17/01 11:41 AM
CLinkedList.hpp 6KB HPP File 9/17/01 11:41 AM
CListFile.h 2KB H File 9/17/01 11:41 AM
CMaterialDB.h 2KB H File 9/17/01 11:41 AM
CMatID.h 3KB H File 9/17/01 11:41 AM
CMSWinDC.h 3KB H File 9/17/01 11:41 AM
CNamed.h 2KB H File 9/17/01 11:41 AM
CNEList.h 2KB H File 9/17/01 11:41 AM
CNEList.hpp 4KB HPP File 9/17/01 11:41 AM
CNewtonRaphson.h 2KB H File 9/17/01 11:41 AM
CNTable.h 3KB H File 9/17/01 11:41 AM
COpenGLDC.h 5KB H File 9/21/01 7:04 AM
COptimize1D.h 3KB H File 9/17/01 11:41 AM
117 object(s) 288KB (Disk free space: 0 bytes) My Computer

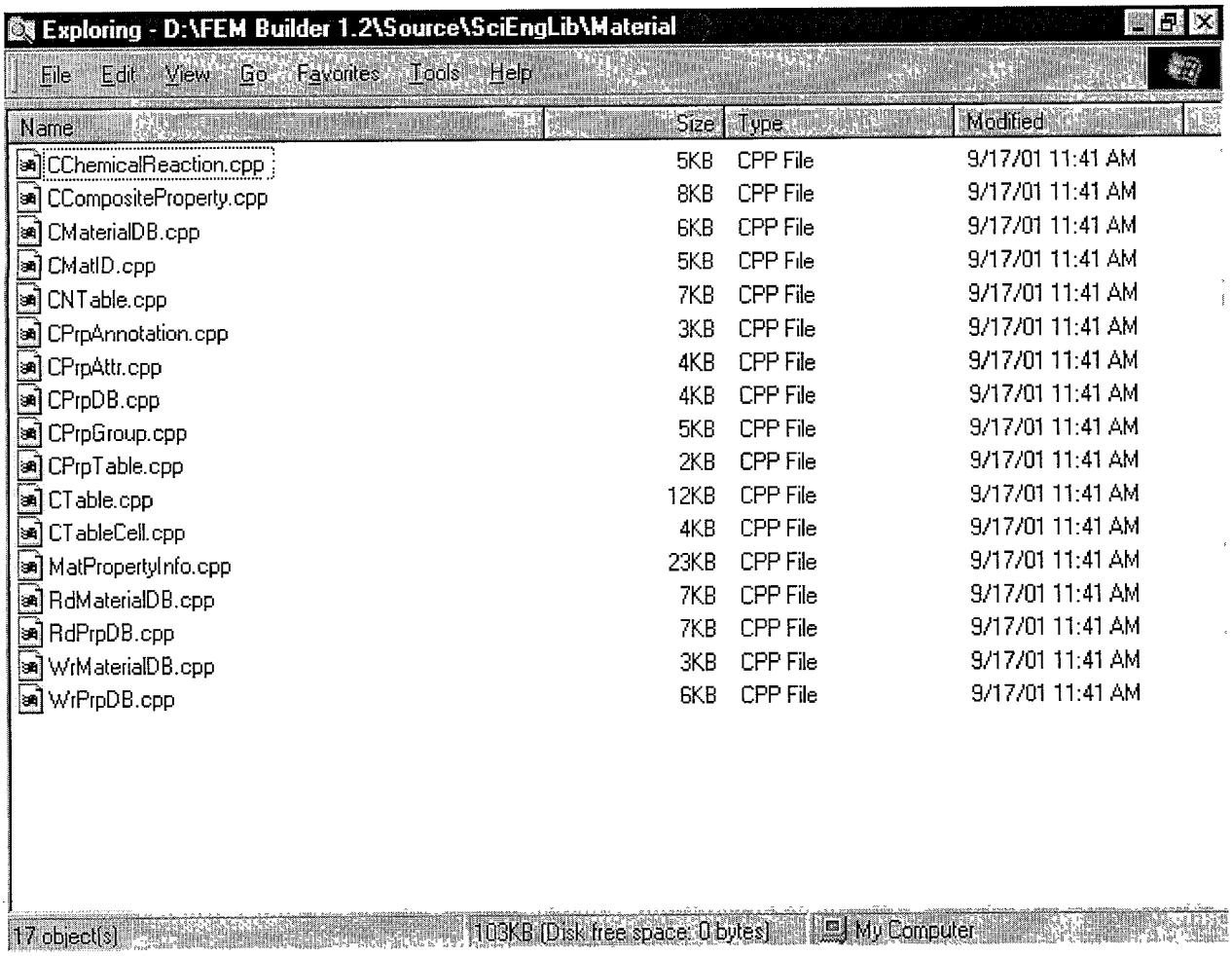

Exploring - D:\FEM Builder 1.2\Source\SciEngLib\Include
File Edit View Go Favorites Tools Help
Name Size Type Modified
COptimizeND.h 3KB H File 9/17/01 11:41 AM
CPrpAnnotation.h 2KB H File 9/17/01 11:41 AM
CPrpAttr.h 2KB H File 9/17/01 11:41 AM
CPrpDB.h 3KB H File 9/17/01 11:41 AM
CPrpGroup.h 2KB H File 9/17/01 11:41 AM
CPrpNote.h 1KB H File 9/17/01 11:41 AM
CPrpTable.h 2KB H File 9/17/01 11:41 AM
CPt.h 2KB H File 9/17/01 11:41 AM
CRelate.h 2KB H File 9/17/01 11:41 AM
CRelate.hpp 1KB HPP File 9/17/01 11:41 AM
CRubberBand.h 2KB H File 9/17/01 11:41 AM
CSet.h 2KB H File 9/17/01 11:41 AM
CSet.hpp 6KB HPP File 9/17/01 11:41 AM
CSetArray.h 6KB H File 9/17/01 11:41 AM
CSpline.h 2KB H File 9/17/01 11:41 AM
CSurface.h 11KB H File 9/17/01 11:41 AM
CSurfaceEdge.h 2KB H File 9/17/01 11:41 AM
CSurfaceFit.h 3KB H File 9/17/01 11:41 AM
CSurfacePatch.h 3KB H File 9/17/01 11:41 AM
CSystem.h 3KB H File 9/17/01 11:41 AM
CTable.h 4KB H File 9/17/01 11:41 AM
CTableCell.h 3KB H File 9/17/01 11:41 AM
CText.h 3KB H File 9/17/01 11:41 AM
117 object(s) 288KB (Disk free space: 0 bytes) My Computer

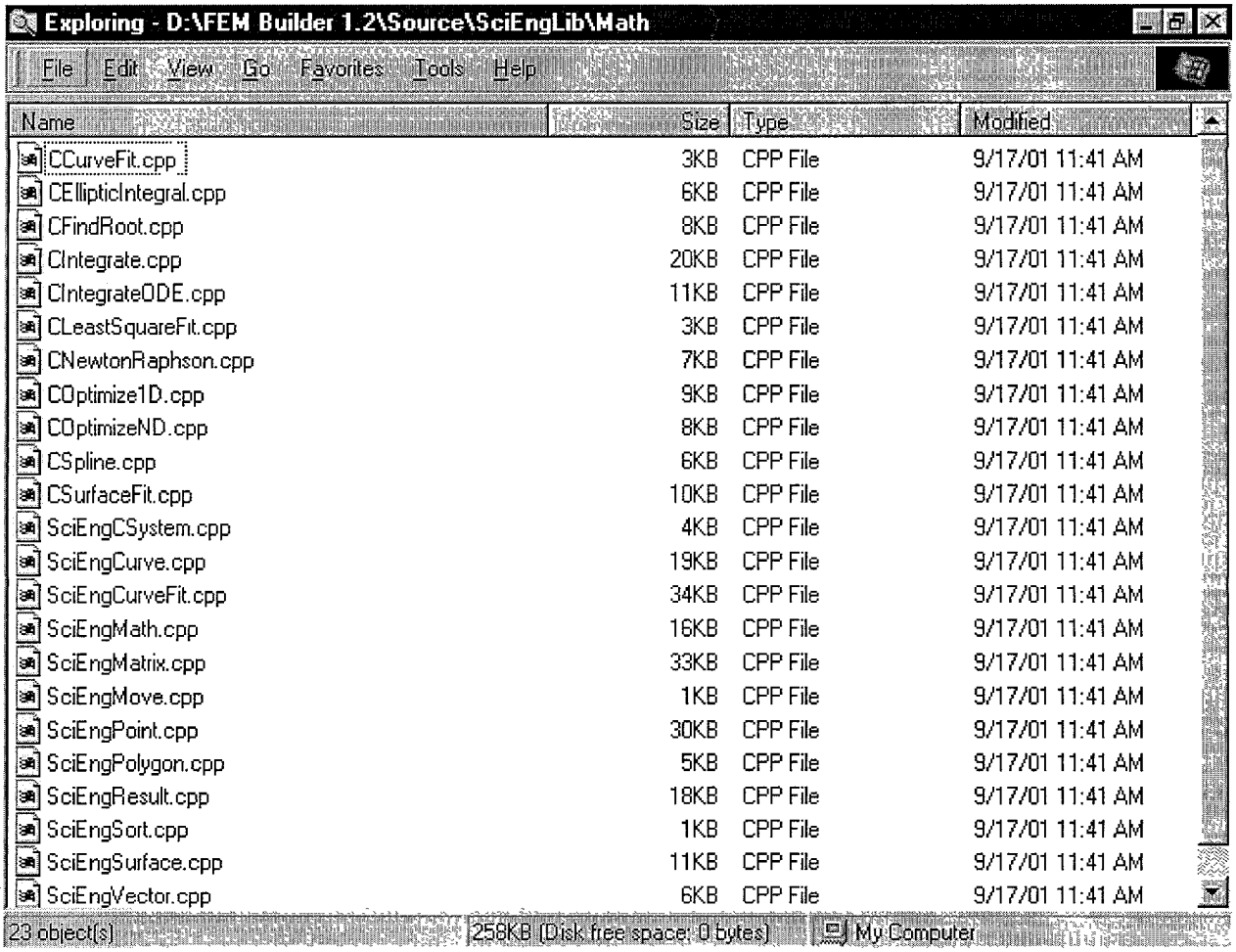

Exploring - D:\FEM Builder 1.2\Source\SciEngLib\Include
File Edit View Go Favorites Tools Help
Name Size Type Modified
CTextFont.h 2KB H File 9/17/01 11:41 AM
CTextFontDialog.h 1KB H File 9/17/01 11:41 AM
CTimeDate.h 2KB H File 9/17/01 11:41 AM
CTimeMS.h 2KB H File 9/17/01 11:41 AM
CUnitSystem.h 4KB H File 9/17/01 11:41 AM
CViewDC.h 10KB H File 9/21/01 7:04 AM
CViewDCDef.h 2KB H File 9/17/01 11:41 AM
CVolume.h 4KB H File 9/17/01 11:41 AM
CWorkPlane.h 4KB H File 9/17/01 11:41 AM
CXYPlotAxis.h 3KB H File 9/17/01 11:41 AM
CXYPlotCurve.h 5KB H File 9/17/01 11:41 AM
CXYPlotData.h 3KB H File 9/21/01 7:04 AM
DrawSymbol.h 2KB H File 9/17/01 11:41 AM
MatProperty.h 2KB H File 9/17/01 11:41 AM
MatPropertyInfo.h 2KB H File 9/17/01 11:41 AM
RdFortranBinary.h 3KB H File 9/17/01 11:41 AM
RdMaterialDB.h 2KB H File 9/17/01 11:41 AM
RdPrpDB.h 2KB H File 9/17/01 11:41 AM
RdTableFile.h 2KB H File 9/17/01 11:41 AM
SciEngBit.h 1KB H File 9/17/01 11:41 AM
SciEngBit.hpp 1KB HPP File 9/17/01 11:41 AM
SciEngColor.h 3KB H File 9/17/01 11:41 AM
SciEngCSystem.h 2KB H File 9/17/01 11:41 AM
117 object(s) 288KB (Disk free space: 0 bytes) My Computer

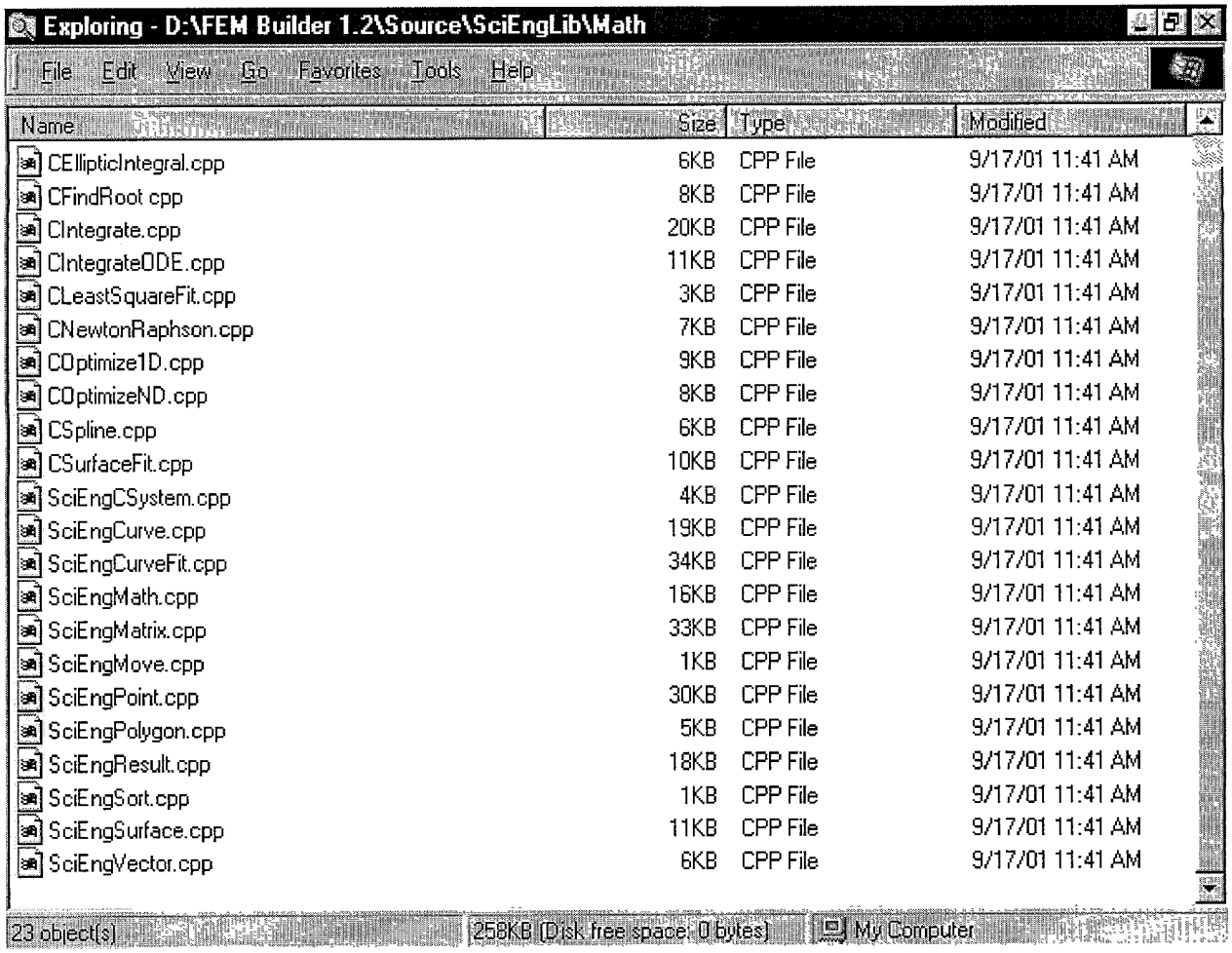

Exploring - D:\FEM Builder 1.2\Source\SciEngLib\Include
File Edit View Go Favorites Tools Help
Name Size Type Modified
SciEngCurve.h 3KB H File 9/17/01 11:41 AM
SciEngCurveFit.h 7KB H File 9/17/01 11:41 AM
SciEngDraw.h 1KB H File 9/17/01 11:41 AM
SciEngError.h 1KB H File 9/17/01 11:41 AM
SciEngFile.h 2KB H File 9/17/01 11:41 AM
SciEngLib.h 3KB H File 9/17/01 11:41 AM
SciEngMath.h 4KB H File 9/17/01 11:41 AM
SciEngMath.hpp 4KB HPP File 9/17/01 11:41 AM
SciEngMatrix.h 3KB H File 9/17/01 11:41 AM
SciEngMove.h 1KB H File 9/17/01 11:41 AM
SciEngMove.hpp 1KB HPP File 9/17/01 11:41 AM
SciEngPoint.h 4KB H File 9/17/01 11:41 AM
SciEngPolygon.h 1KB H File 9/17/01 11:41 AM
SciEngResult.h 2KB H File 9/17/01 11:41 AM
SciEngSort.h 1KB H File 9/17/01 11:41 AM
SciEngSort.hpp 5KB HPP File 9/17/01 11:41 AM
SciEngStddfn.h 3KB H File 9/17/01 11:41 AM
SciEngStr.h 3KB H File 9/17/01 11:41 AM
SciEngSurface.h 1KB H File 9/17/01 11:41 AM
SciEngVector.h 2KB H File 9/17/01 11:41 AM
SciEngVector.hpp 1KB HPP File 9/17/01 11:41 AM
StdAfx.h 1KB H File 9/17/01 11:41 AM
WrMaterialDB.h 1KB H File 9/17/01 11:41 AM
117 object(s) 288KB (Disk free space: 0 bytes) My Computer

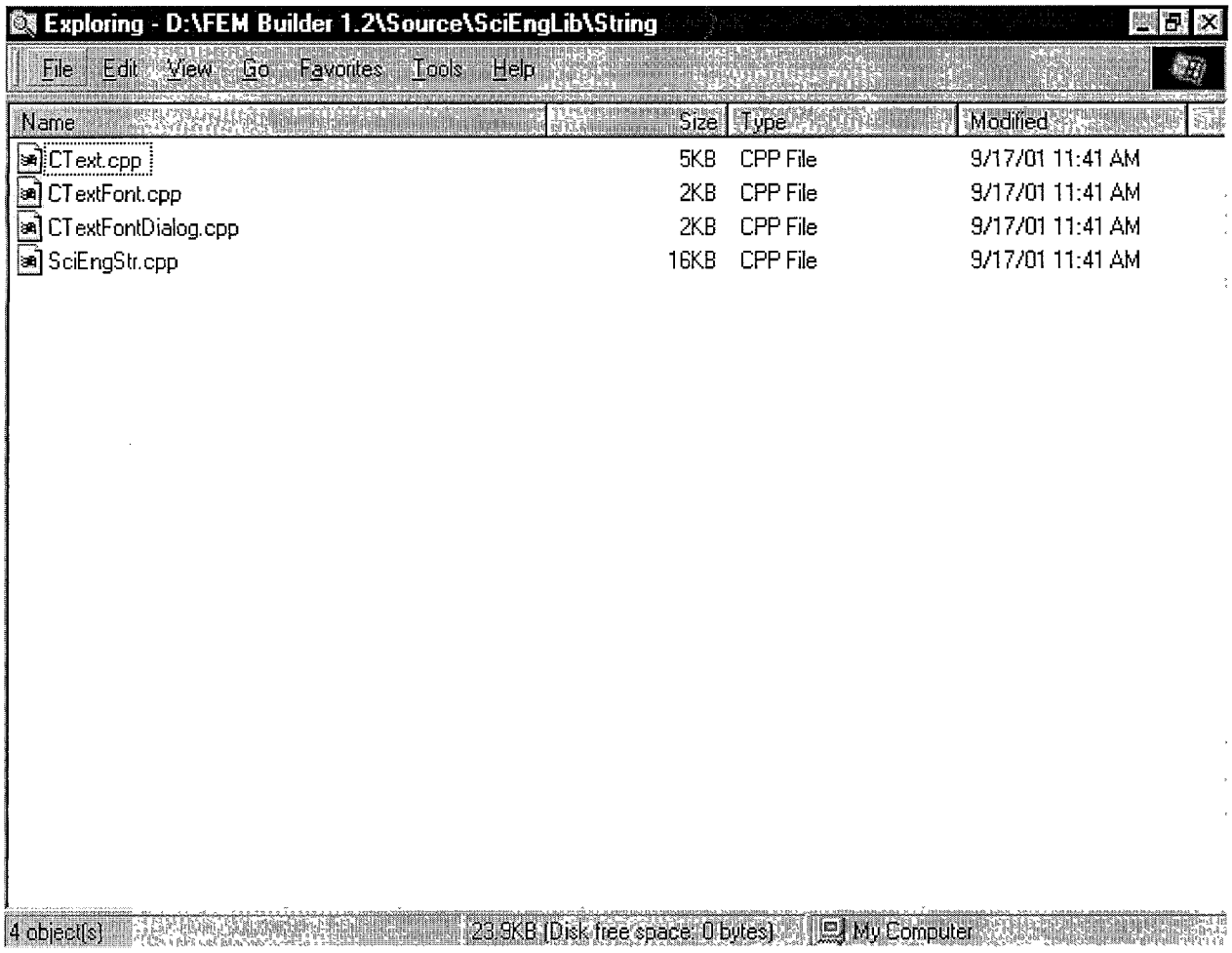

Exploring - D:\FEM Builder 1.2\Source\SciEngLib\Include
File Edit View Go Favorites Tools Help
Name Size Type Modified
SciEngError.h 1KB H File 9/17/01 11:41 AM
SciEngFile.h 2KB H File 9/17/01 11:41 AM
SciEngLib.h 3KB H File 9/17/01 11:41 AM
SciEngMath h 4KB H File 9/17/01 11:41 AM
SciEngMath.hpp 4KB HPP File 9/17/01 11:41 AM
SciEngMatrix.h 3KB H File 9/17/01 11:41 AM
SciEngMove.h 1KB H File 9/17/01 11:41 AM
SciEngMove.hpp 1KB HPP File 9/17/01 11:41 AM
SciEngPoint.h 4KB H File 9/17/01 11:41 AM
SciEngPolygon.h 1KB H File 9/17/01 11:41 AM
SciEngResult.h 2KB H File 9/17/01 11:41 AM
SciEngSort.h 1KB H File 9/17/01 11:41 AM
SciEngSort.hpp 5KB HPP File 9/17/01 11:41 AM
SciEngStddfn.h 3KB H File 9/17/01 11:41 AM
SciEngStr.h 3KB H File 9/17/01 11:41 AM
SciEngSurface.h 1KB H File 9/17/01 11:41 AM
SciEngVector.h 2KB H File 9/17/01 11:41 AM
SciEngVector.hpp 1KB HPP File 9/17/01 11:41 AM
StdAfx.h 1KB H File 9/17/01 11:41 AM
WrMaterialDB.h 1KB H File 9/17/01 11:41 AM
WrPrpDB.h 2KB H File 9/17/01 11:41 AM
WrTableFile.h 1KB H File 9/17/01 11:41 AM
117 object(s) 288KB (Disk free space: 0 bytes) My Computer

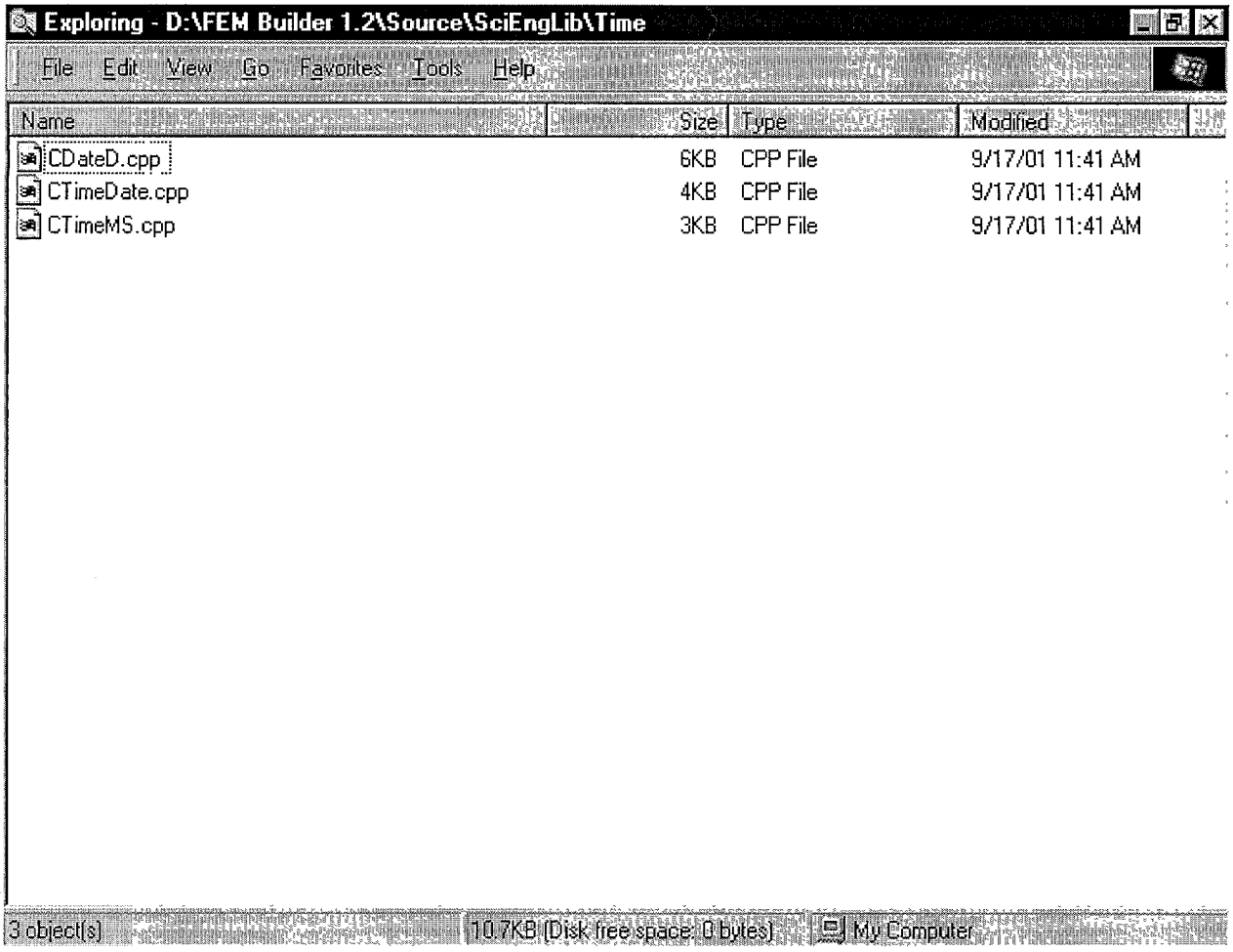

Exploring - D:\FEM Builder 1.2\Source\SciEngLib\List
File Edit View Go Favorites Tools Help
Name Size Type Modified
CDoubleList.cpp 3KB CPP File 9/17/01 11:41 AM
CHash.cpp 17KB CPP File 9/17/01 11:41 AM
CIntList.cpp 4KB CPP File 9/17/01 11:41 AM
3 object(s) 21.4KB (Disk free space: 0 bytes) My Computer

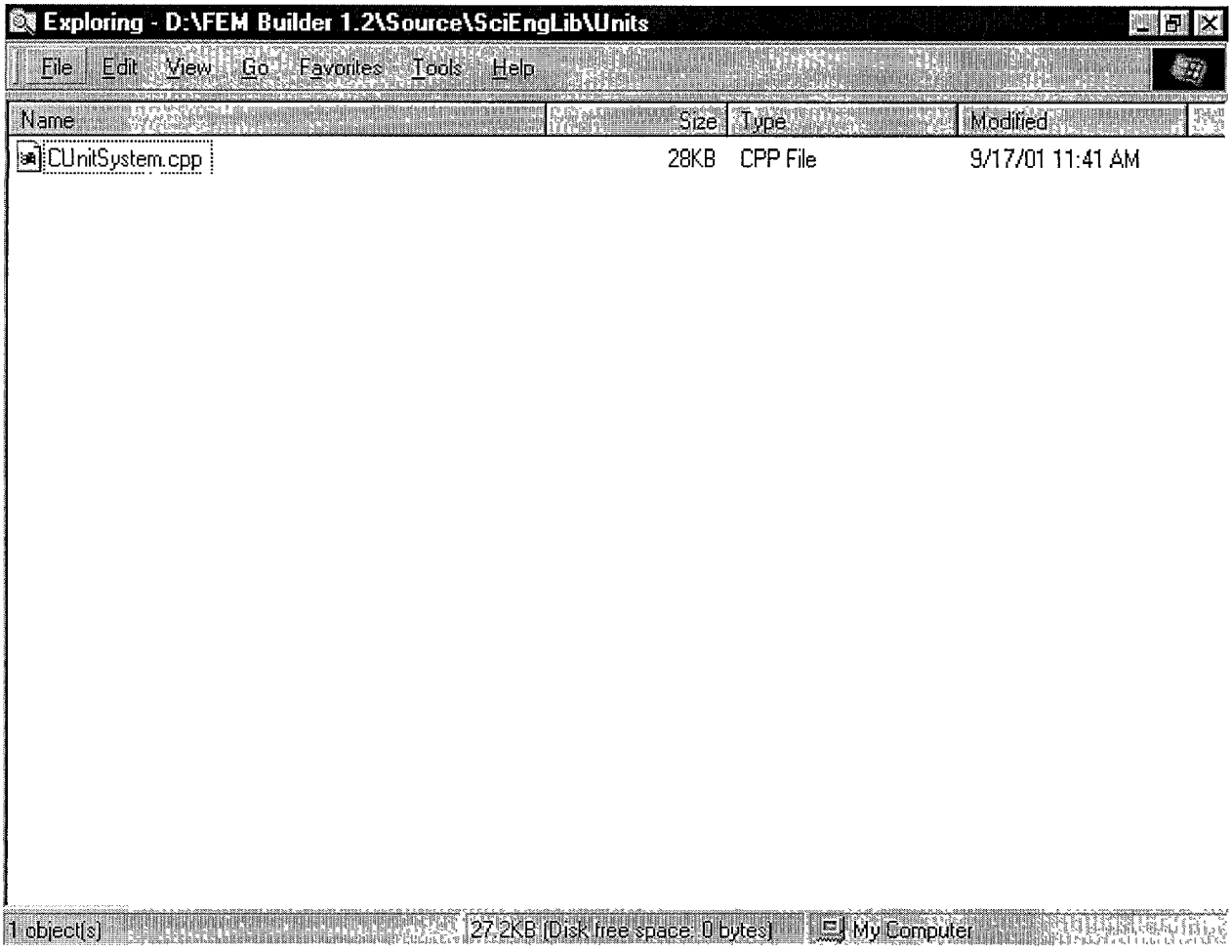

Exploring - D:\FEM Builder 1.2\Source\SciEngLib\Material
File Edit View Go Favorites Tools Help
Name Size Type Modified
CChemicalReaction.cpp 5KB CPP File 9/17/01 11:41 AM
CCompositeProperty.cpp 8KB CPP File 9/17/01 11:41 AM
CMaterialDB.cpp 6KB CPP File 9/17/01 11:41 AM
CMatID.cpp 5KB CPP File 9/17/01 11:41 AM
CNTable.cpp 7KB CPP File 9/17/01 11:41 AM
CPrpAnnotation.cpp 3KB CPP File 9/17/01 11:41 AM
CPrpAttr.cpp 4KB CPP File 9/17/01 11:41 AM
CPrpDB.cpp 4KB CPP File 9/17/01 11:41 AM
CPrpGroup.cpp 5KB CPP File 9/17/01 11:41 AM
CPrpTable.cpp 2KB CPP File 9/17/01 11:41 AM
CTable.cpp 12KB CPP File 9/17/01 11:41 AM
CTableCell.cpp 4KB CPP File 9/17/01 11:41 AM
MatPropertyInfo.cpp 23KB CPP File 9/17/01 11:41 AM
RdMaterialDB.cpp 7KB CPP File 9/17/01 11:41 AM
RdPrpDB.cpp 7KB CPP File 9/17/01 11:41 AM
WrMaterialDB.cpp 3KB CPP File 9/17/01 11:41 AM
WrPrpDB.cpp 6KB CPP File 9/17/01 11:41 AM
17 object(s) 103KB (Disk free space: 0 bytes) My Computer

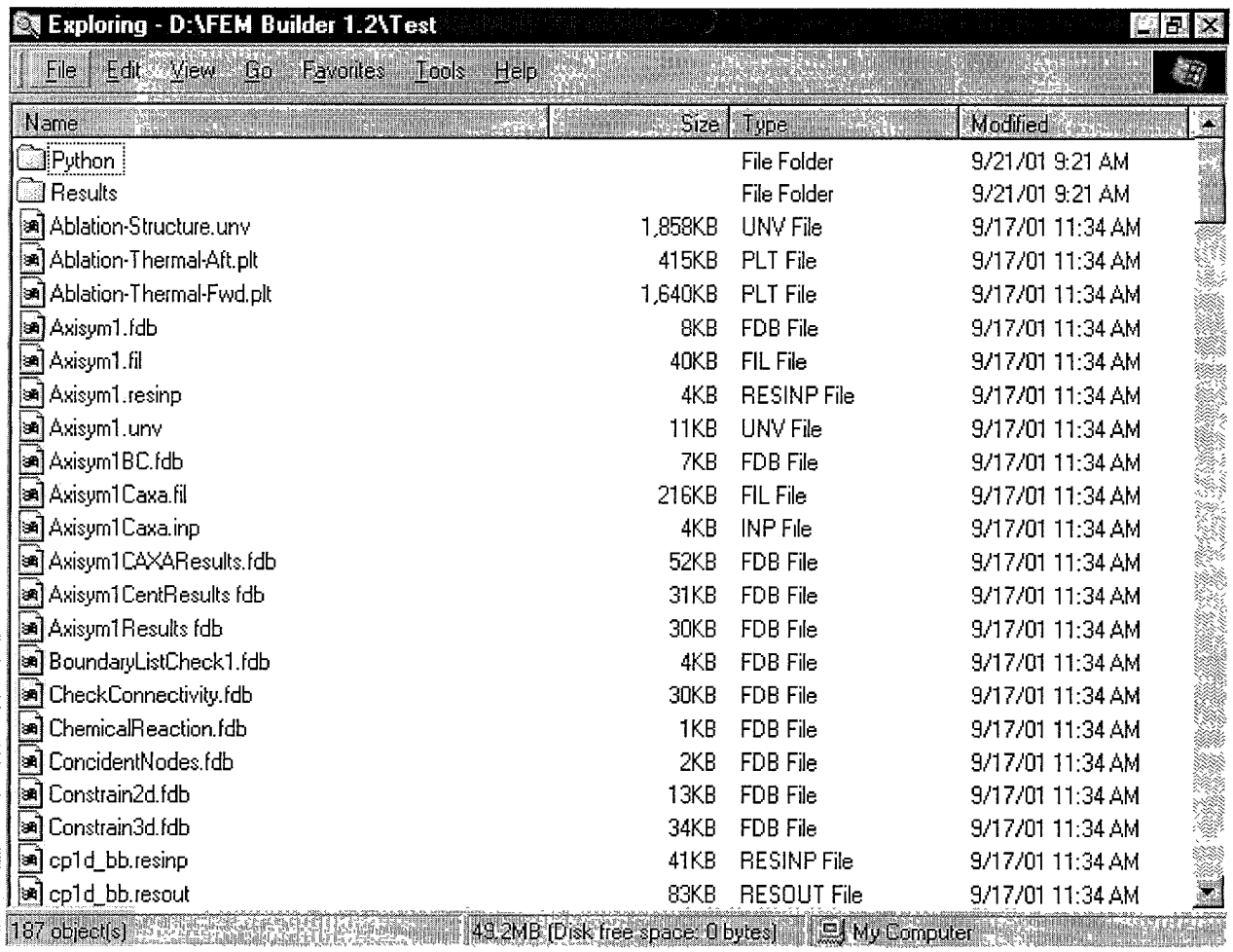

Exploring - D:\FEM Builder 1.2\Source\SciEngLib\Math
File Edit View Go Favorites Tools Help
Name Size Type Modified
CCurveFit.cpp 3KB CPP File 9/17/01 11:41 AM
CEllipticIntegral.cpp 6KB CPP File 9/17/01 11:41 AM
CFindRoot.cpp 8KB CPP File 9/17/01 11:41 AM
CIntegrate.cpp 20KB CPP File 9/17/01 11:41 AM
CIntegrateODE.cpp 11KB CPP File 9/17/01 11:41 AM
CLeastSquareFit.cpp 3KB CPP File 9/17/01 11:41 AM
CNewtonRaphson.cpp 7KB CPP File 9/17/01 11:41 AM
COptimize1D.cpp 9KB CPP File 9/17/01 11:41 AM
COptimizeND.cpp 8KB CPP File 9/17/01 11:41 AM
CSpline.cpp 6KB CPP File 9/17/01 11:41 AM
CSurfaceFit.cpp 10KB CPP File 9/17/01 11:41 AM
SciEngCSystem.cpp 4KB CPP File 9/17/01 11:41 AM
SciEngCurve.cpp 19KB CPP File 9/17/01 11:41 AM
SciEngCurveFit.cpp 34KB CPP File 9/17/01 11:41 AM
SciEngMath.cpp 16KB CPP File 9/17/01 11:41 AM
SciEngMatrix.cpp 33KB CPP File 9/17/01 11:41 AM
SciEngMove.cpp 1KB CPP File 9/17/01 11:41 AM
SciEngPoint.cpp 30KB CPP File 9/17/01 11:41 AM
SciEngPolygon.cpp 5KB CPP File 9/17/01 11:41 AM
SciEngResult.cpp 18KB CPP File 9/17/01 11:41 AM
SciEngSort.cpp 1KB CPP File 9/17/01 11:41 AM
SciEngSurface.cpp 11KB CPP File 9/17/01 11:41 AM
SciEngVector.cpp 6KB CPP File 9/17/01 11:41 AM
23 object(s) 258KB (Disk free space: 0 bytes) My Computer

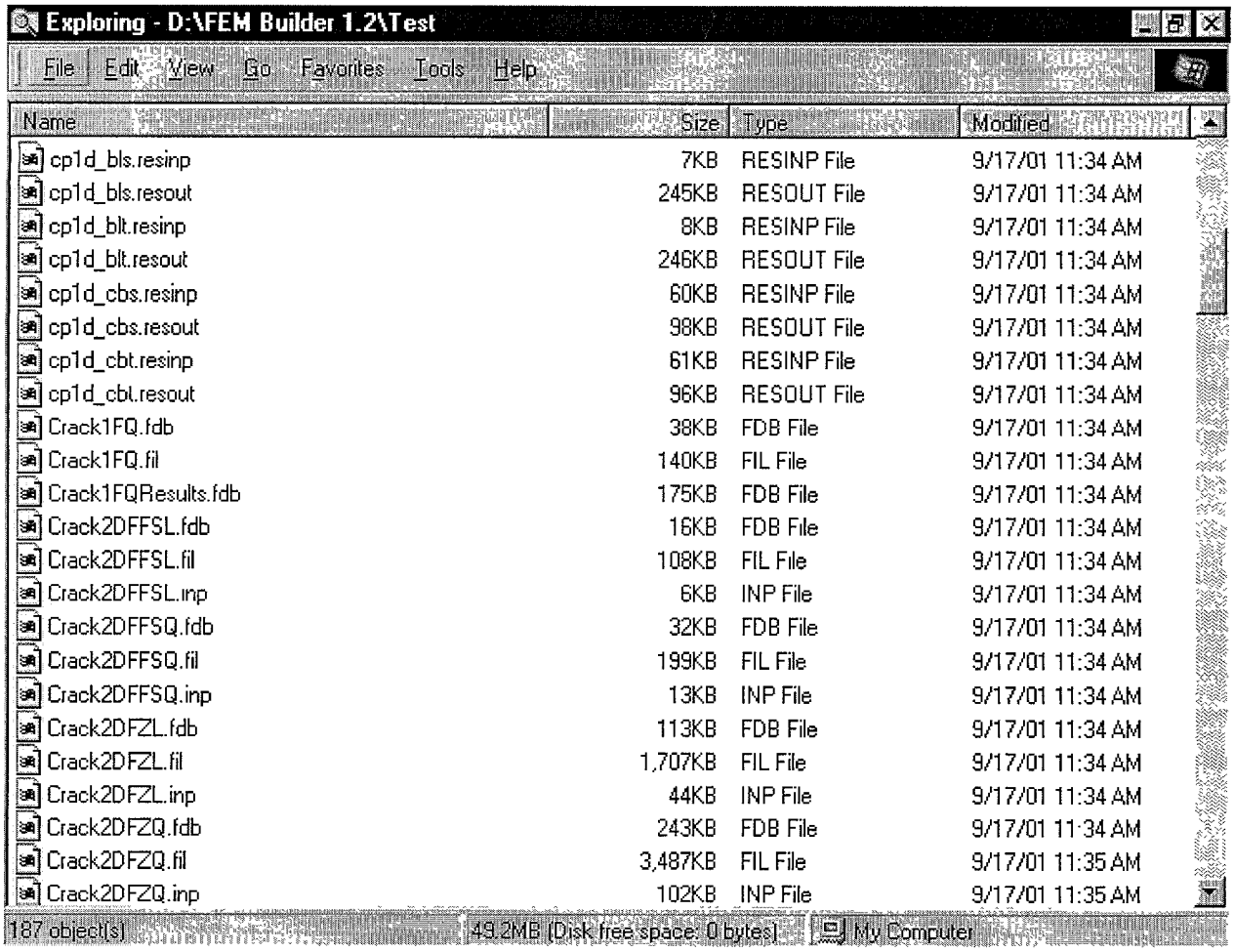

Exploring - D:\FEM Builder 1.2\Source\SciEngLib\Math
File Edit View Go Favorites Tools Help
Name Size Type Modified
CEllipticIntegral.cpp 6KB CPP File 9/17/01 11:41 AM
CFindRoot.cpp 8KB CPP File 9/17/01 11:41 AM
CIntegrate.cpp 20KB CPP File 9/17/01 11:41 AM
CIntegrateODE.cpp 11KB CPP File 9/17/01 11:41 AM
CLeastSquareFit.cpp 3KB CPP File 9/17/01 11:41 AM
CNewtonRaphson.cpp 7KB CPP File 9/17/01 11:41 AM
COptimize1D.cpp 9KB CPP File 9/17/01 11:41 AM
COptimizeND.cpp 8KB CPP File 9/17/01 11:41 AM
CSpline.cpp 6KB CPP File 9/17/01 11:41 AM
CSurfaceFit.cpp 10KB CPP File 9/17/01 11:41 AM
SciEngCSystem.cpp 4KB CPP File 9/17/01 11:41 AM
SciEngCurve.cpp 19KB CPP File 9/17/01 11:41 AM
SciEngCurveFit.cpp 34KB CPP File 9/17/01 11:41 AM
SciEngMath.cpp 16KB CPP File 9/17/01 11:41 AM
SciEngMatrix.cpp 33KB CPP File 9/17/01 11:41 AM
SciEngMove.cpp 1KB CPP File 9/17/01 11:41 AM
SciEngPoint.cpp 30KB CPP File 9/17/01 11:41 AM
SciEngPolygon.cpp 5KB CPP File 9/17/01 11:41 AM
SciEngResult.cpp 18KB CPP File 9/17/01 11:41 AM
SciEngSort.cpp 1KB CPP File 9/17/01 11:41 AM
SciEngSurface.cpp 11KB CPP File 9/17/01 11:41 AM
SciEngVector.cpp 6KB CPP File 9/17/01 11:41 AM
23 object(s) 258KB (Disk free space: 0 bytes) My Computer

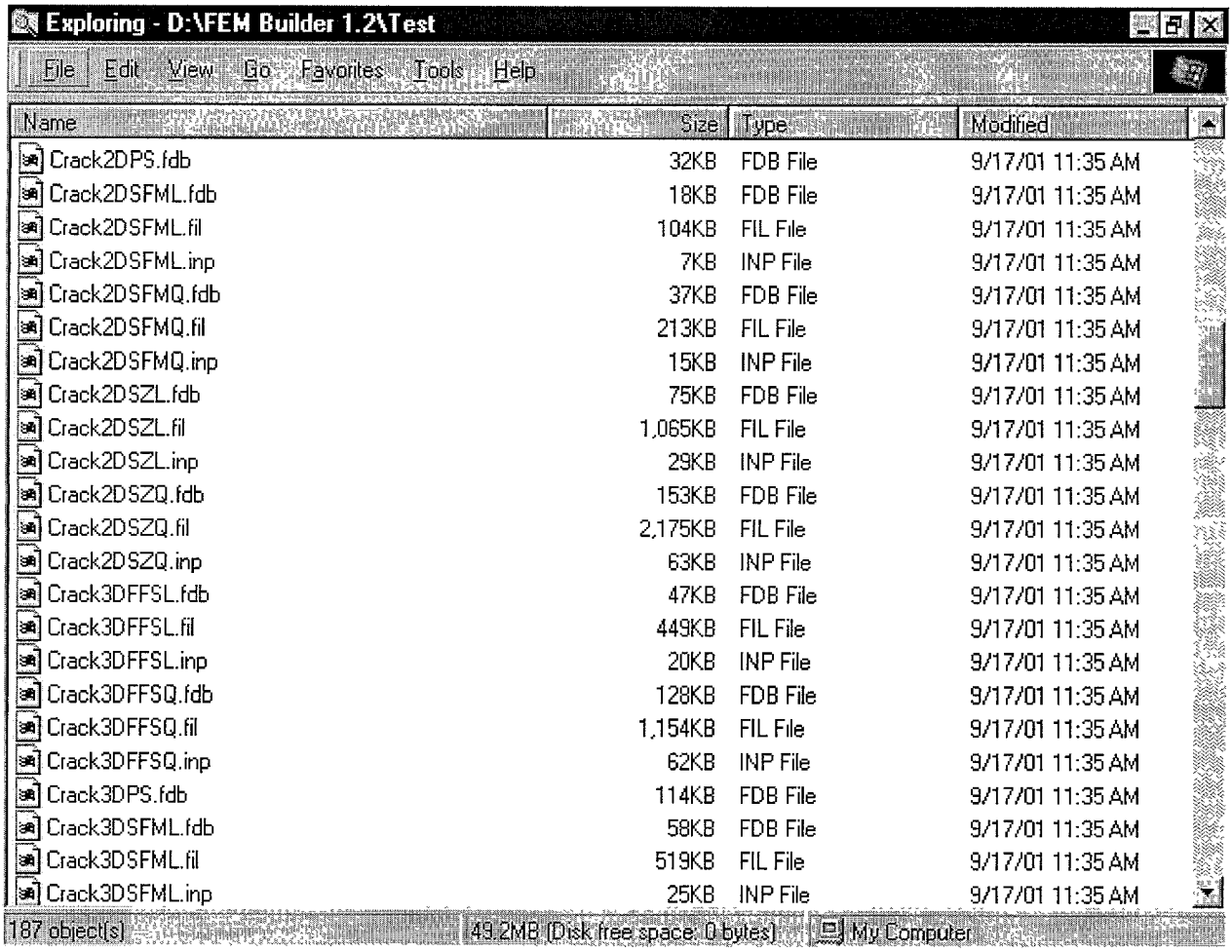

Exploring - D:\FEM Builder 1.2\Source\SciEngLib\String
File Edit View Go Favorites Tools Help
Name Size Type Modified
CText.cpp 5KB CPP File 9/17/01 11:41 AM
CTextFont.cpp 2KB CPP File 9/17/01 11:41 AM
CTextFontDialog.cpp 2KB CPP File 9/17/01 11:41 AM
SciEngStr.cpp 16KB CPP File 9/17/01 11:41 AM
4 object(s)
23.9KB (Disk free space: 0 bytes)
My Computer

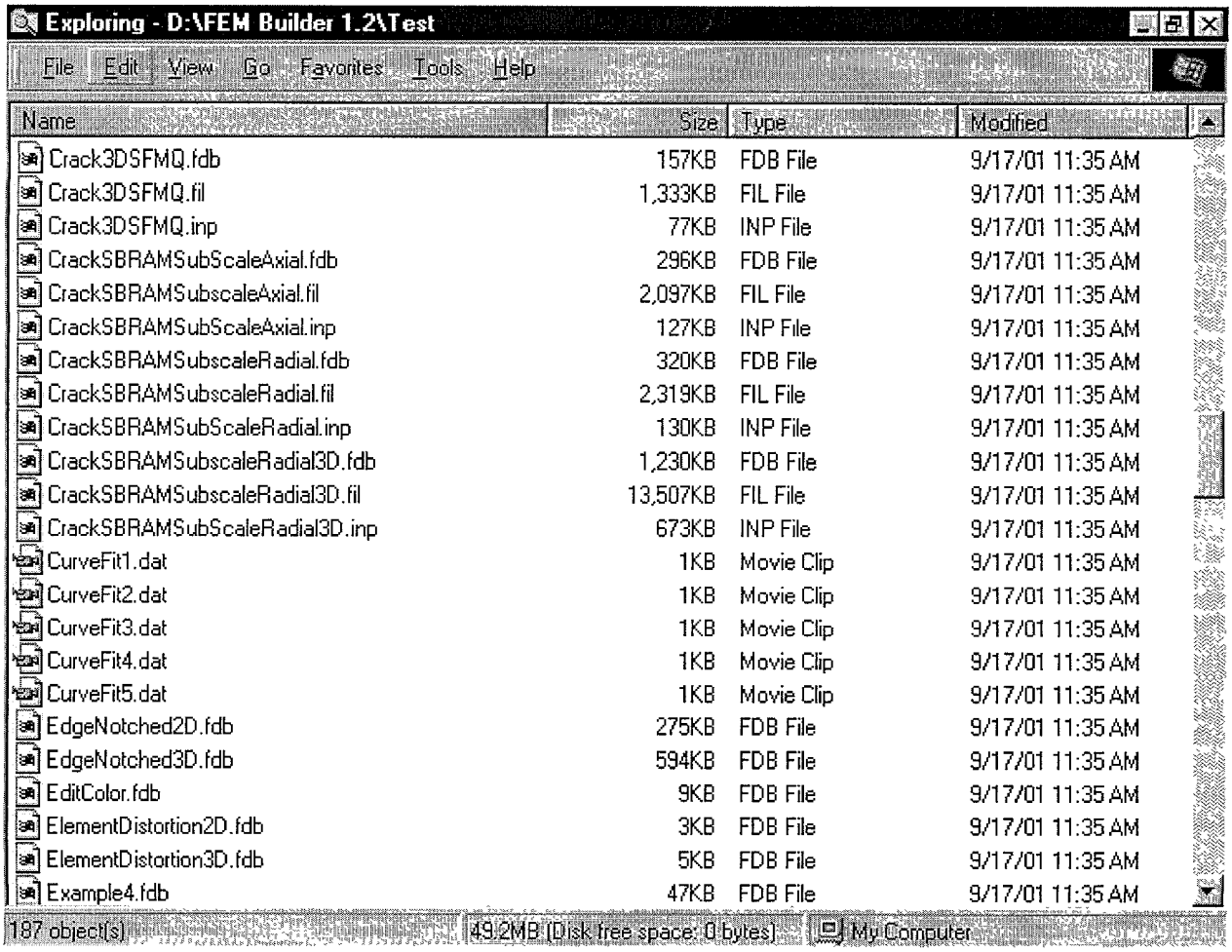

Exploring - D:\FEM Builder 1.2\Source\SciEngLib\Time
File Edit View Go Favorites Tools Help
Name Size Type Modified
CDateD.cpp 6KB CPP File 9/17/01 11:41 AM
CTimeDate.cpp 4KB CPP File 9/17/01 11:41 AM
CTimeMS.cpp 3KB CPP File 9/17/01 11:41 AM
3 object(s) 10.7KB (Disk free space: 0 bytes) My Computer

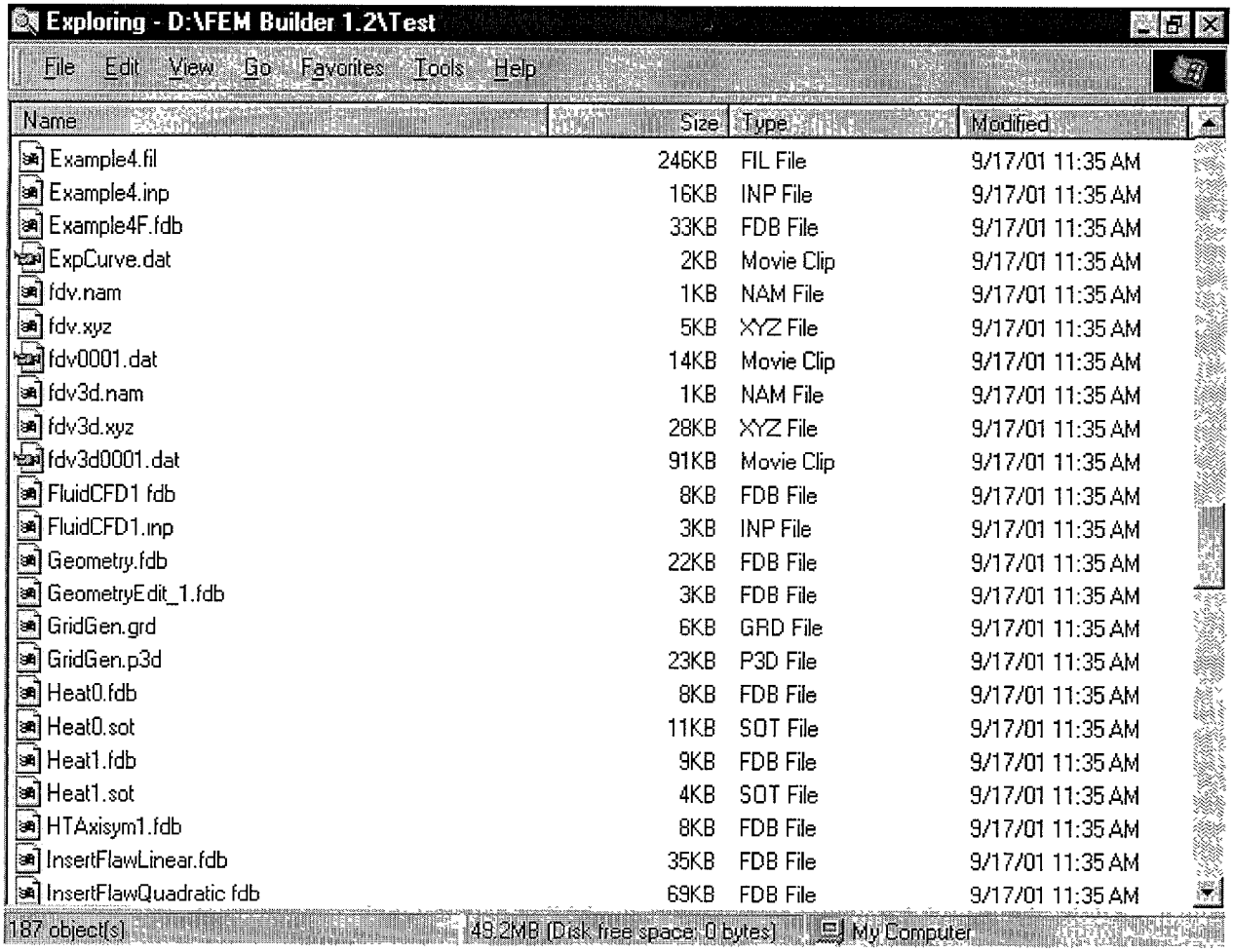

Exploring - D:\FEM Builder 1.2\Source\SciEngLib\Units
File Edit View Go Favorites Tools Help
Name
Size
Type
Modified
CUnitSystem.cpp
28KB
CPP File
9/17/01 11:41 AM
1 object(s)
27.2KB (Disk free space: 0 bytes)
My Computer

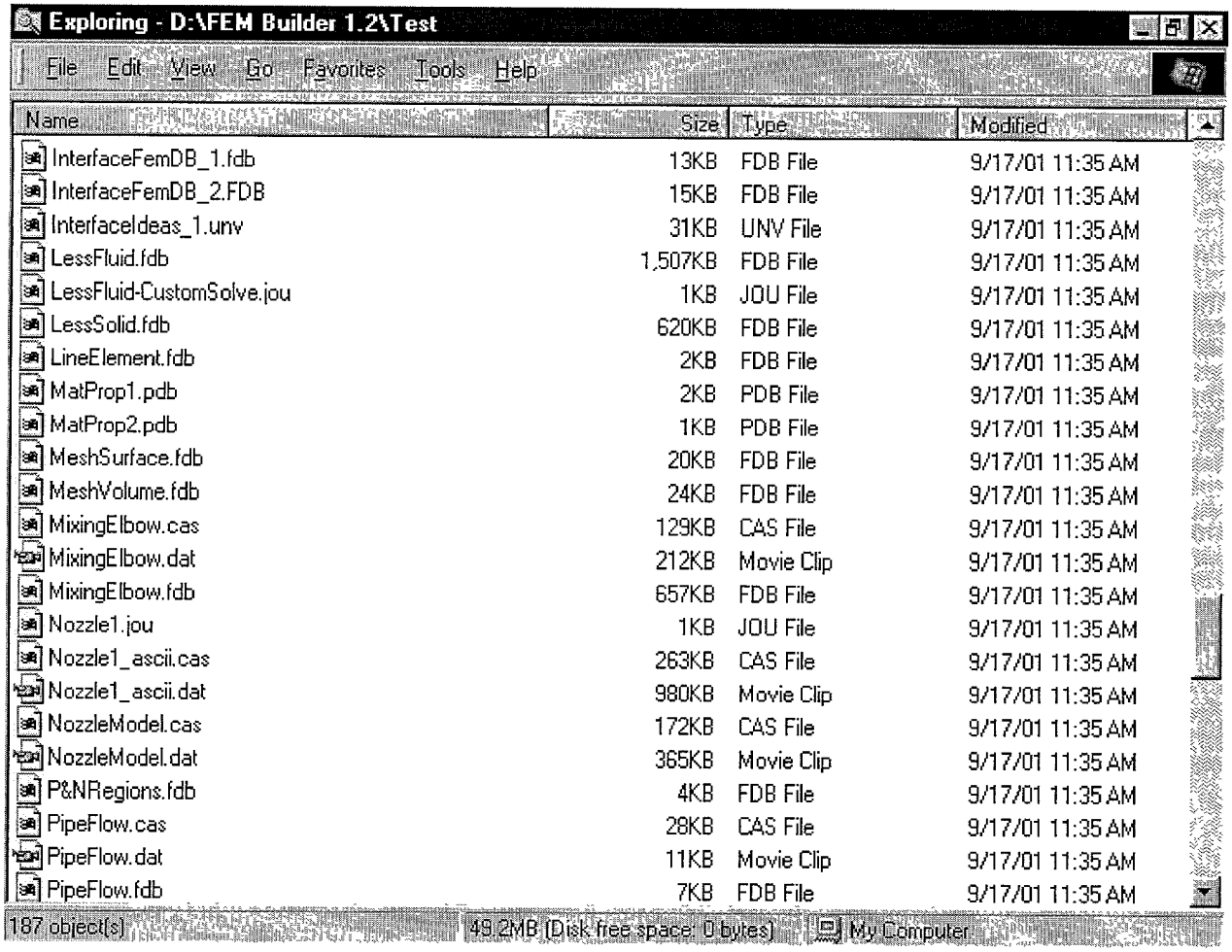

Exploring - D:\FEM Builder 1.2\Test
File Edit View Go Favorites Tools Help
Name Size Type Modified
Python File Folder 9/21/01 9:21 AM
Results File Folder 9/21/01 9:21 AM
Ablation-Structure.unv 1,858KB UNV File 9/17/01 11:34 AM
Ablation-Thermal-Aft.plt 415KB PLT File 9/17/01 11:34 AM
Ablation-Thermal-Fwd.plt 1,640KB PLT File 9/17/01 11:34 AM
Axisym1.fdb 8KB FDB File 9/17/01 11:34 AM
Axisym1.fil 40KB FIL File 9/17/01 11:34 AM
Axisym1.resinp 4KB RESINP File 9/17/01 11:34 AM
Axisym1.unv 11KB UNV File 9/17/01 11:34 AM
Axisym1BC.fdb 7KB FDB File 9/17/01 11:34 AM
Axisym1Caxa.fil 216KB FIL File 9/17/01 11:34 AM
Axisym1Caxa.inp 4KB INP File 9/17/01 11:34 AM
Axisym1CAXAResults.fdb 52KB FDB File 9/17/01 11:34 AM
Axisym1CentResults.fdb 31KB FDB File 9/17/01 11:34 AM
Axisym1Results.fdb 30KB FDB File 9/17/01 11:34 AM
BoundaryListCheck1.fdb 4KB FDB File 9/17/01 11:34 AM
CheckConnectivity.fdb 30KB FDB File 9/17/01 11:34 AM
ChemicalReaction.fdb 1KB FDB File 9/17/01 11:34 AM
ConcidentNodes.fdb 2KB FDB File 9/17/01 11:34 AM
Constrain2d.fdb 13KB FDB File 9/17/01 11:34 AM
Constrain3d.fdb 34KB FDB File 9/17/01 11:34 AM
cp1d_bb.resinp 41KB RESINP File 9/17/01 11:34 AM
cp1d_bb.resout 83KB RESOUT File 9/17/01 11:34 AM
187 object(s) 49.2MB (Disk free space: 0 bytes) My Computer

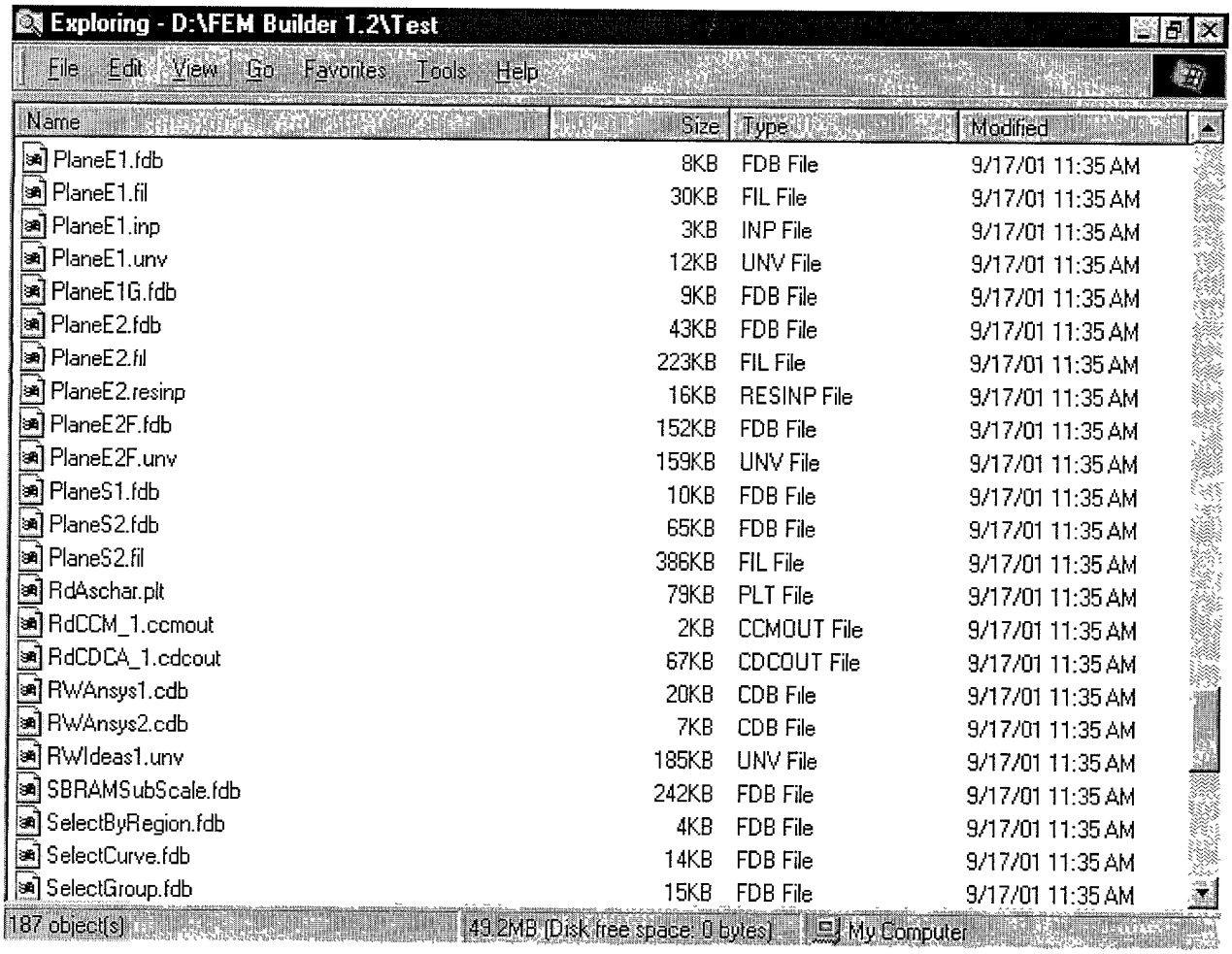

Exploring - D:\FEM Builder 1.2\Test
File Edit View Go Favorites Tools Help
Name Size Type Modified
cp1d_bls.resinp 7KB RESINP File 9/17/01 11:34 AM
cp1d_bls.resout 245KB RESOUT File 9/17/01 11:34 AM
cp1d_blt.resinp 8KB RESINP File 9/17/01 11:34 AM
cp1d_blt.resout 246KB RESOUT File 9/17/01 11:34 AM
cp1d_cbs.resinp 60KB RESINP File 9/17/01 11:34 AM
cp1d_cbs.resout 98KB RESOUT File 9/17/01 11:34 AM
cp1d_cbt.resinp 61KB RESINP File 9/17/01 11:34 AM
cp1d_cbt.resout 96KB RESOUT File 9/17/01 11:34 AM
Crack1FQ.fdb 38KB FDB File 9/17/01 11:34 AM
Crack1FQ.fil 140KB FIL File 9/17/01 11:34 AM
Crack1FQResults.fdb 175KB FDB File 9/17/01 11:34 AM
Crack2DFFSL.fdb 16KB FDB File 9/17/01 11:34 AM
Crack2DFFSL.fil 108KB FIL File 9/17/01 11:34 AM
Crack2DFFSL.inp 6KB INP File 9/17/01 11:34 AM
Crack2DFFSQ.fdb 32KB FDB File 9/17/01 11:34 AM
Crack2DFFSQ.fil 199KB FIL File 9/17/01 11:34 AM
Crack2DFFSQ.inp 13KB INP File 9/17/01 11:34 AM
Crack2DFZL.fdb 113KB FDB File 9/17/01 11:34 AM
Crack2DFZL.fil 1,707KB FIL File 9/17/01 11:34 AM
Crack2DFZL.inp 44KB INP File 9/17/01 11:34 AM
Crack2DFZQ.fdb 243KB FDB File 9/17/01 11:34 AM
Crack2DFZQ.fil 3,487KB FIL File 9/17/01 11:35 AM
Crack2DFZQ.inp 102KB INP File 9/17/01 11:35 AM
187 object(s) 49.2MB (Disk free space: 0 bytes) My Computer

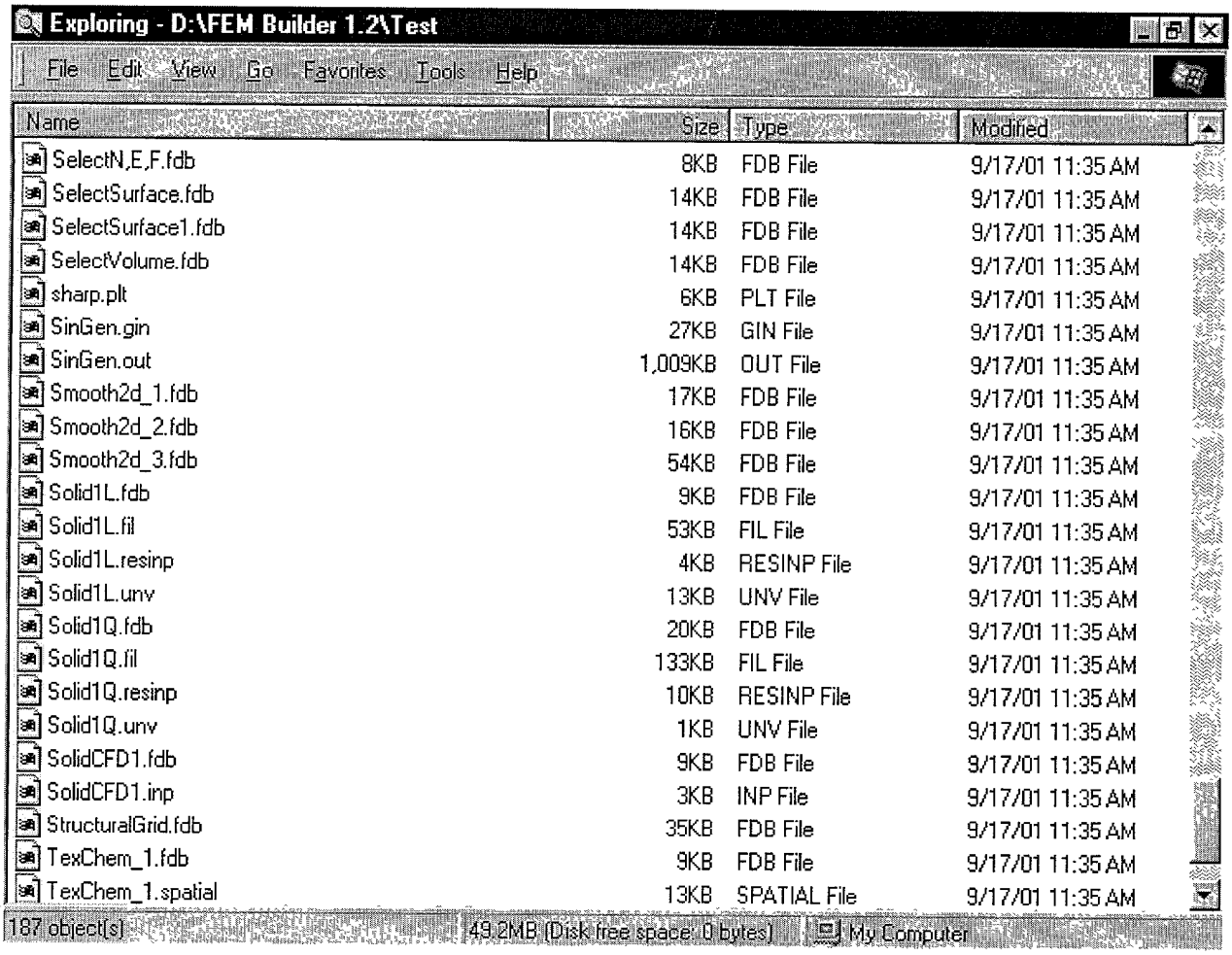

Exploring - D:\FEM Builder 1.2\Test
File Edit View Go Favorites Tools Help
Name Size Type Modified
Crack2DPS.fdb 32KB FDB File 9/17/01 11:35 AM
Crack2DSFML.fdb 18KB FDB File 9/17/01 11:35 AM
Crack2DSFML.fil 104KB FIL File 9/17/01 11:35 AM
Crack2DSFML.inp 7KB INP File 9/17/01 11:35 AM
Crack2DSFMQ.fdb 37KB FDB File 9/17/01 11:35 AM
Crack2DSFMQ.fil 213KB FIL File 9/17/01 11:35 AM
Crack2DSFMQ.inp 15KB INP File 9/17/01 11:35 AM
Crack2DSZL.fdb 75KB FDB File 9/17/01 11:35 AM
Crack2DSZL.fil 1,065KB FIL File 9/17/01 11:35 AM
Crack2DSZL.inp 29KB INP File 9/17/01 11:35 AM
Crack2DSZQ.fdb 153KB FDB File 9/17/01 11:35 AM
Crack2DSZQ.fil 2,175KB FIL File 9/17/01 11:35 AM
Crack2DSZQ.inp 63KB INP File 9/17/01 11:35 AM
Crack3DFFSL.fdb 47KB FDB File 9/17/01 11:35 AM
Crack3DFFSL.fil 449KB FIL File 9/17/01 11:35 AM
Crack3DFFSL.inp 20KB INP File 9/17/01 11:35 AM
Crack3DFFSQ.fdb 128KB FDB File 9/17/01 11:35 AM
Crack3DFFSQ.fil 1,154KB FIL File 9/17/01 11:35 AM
Crack3DFFSQ.inp 62KB INP File 9/17/01 11:35 AM
Crack3DPS.fdb 114KB FDB File 9/17/01 11:35 AM
Crack3DSFML.fdb 58KB FDB File 9/17/01 11:35 AM
Crack3DSFML.fil 519KB FIL File 9/17/01 11:35 AM
Crack3DSFML.inp 25KB INP File 9/17/01 11:35 AM
187 object(s) 49.2MB (Disk free space: 0 bytes) My Computer

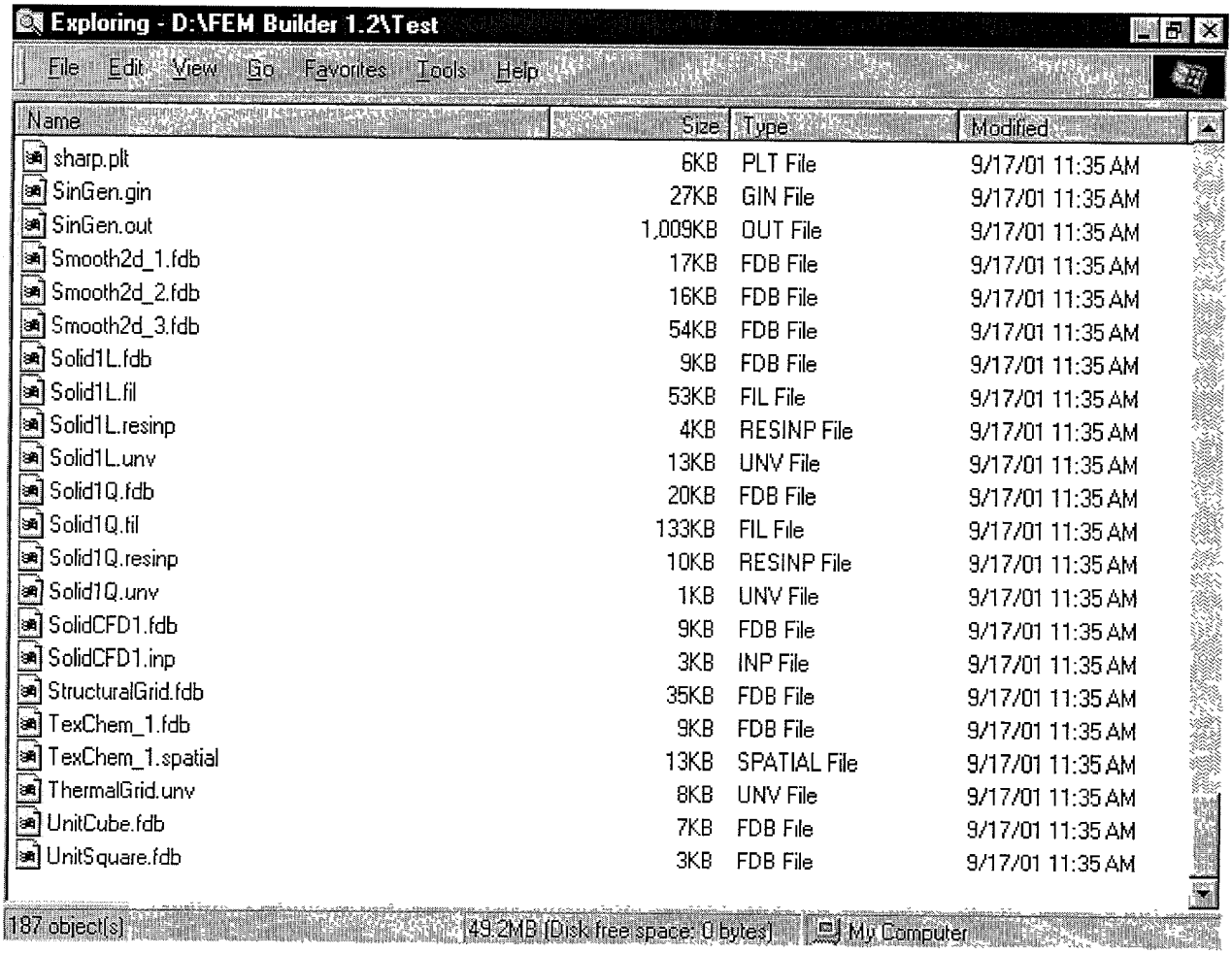

Exploring - D:\FEM Builder 1.2\Test
File Edit View Go Favorites Tools Help
Name Size Type Modified
Crack3DSFMQ.fdb 157KB FDB File 9/17/01 11:35 AM
Crack3DSFMQ.fil 1,333KB FIL File 9/17/01 11:35 AM
Crack3DSFMQ.inp 77KB INP File 9/17/01 11:35 AM
CrackSBRAMSubScaleAxial.fdb 296KB FDB File 9/17/01 11:35 AM
CrackSBRAMSubscaleAxial.fil 2,097KB FIL File 9/17/01 11:35 AM
CrackSBRAMSubScaleAxial.inp 127KB INP File 9/17/01 11:35 AM
CrackSBRAMSubscaleRadial.fdb 320KB FDB File 9/17/01 11:35 AM
CrackSBRAMSubscaleRadial.fil 2,319KB FIL File 9/17/01 11:35 AM
CrackSBRAMSubScaleRadial.inp 130KB INP File 9/17/01 11:35 AM
CrackSBRAMSubscaleRadial3D.fdb 1,230KB FDB File 9/17/01 11:35 AM
CrackSBRAMSubscaleRadial3D.fil 13,507KB FIL File 9/17/01 11:35 AM
CrackSBRAMSubScaleRadial3D.inp 673KB INP File 9/17/01 11:35 AM
CurveFit1.dat 1KB Movie Clip 9/17/01 11:35 AM
CurveFit2.dat 1KB Movie Clip 9/17/01 11:35 AM
CurveFit3.dat 1KB Movie Clip 9/17/01 11:35 AM
CurveFit4.dat 1KB Movie Clip 9/17/01 11:35 AM
CurveFit5.dat 1KB Movie Clip 9/17/01 11:35 AM
EdgeNotched2D.fdb 275KB FDB File 9/17/01 11:35 AM
EdgeNotched3D.fdb 594KB FDB File 9/17/01 11:35 AM
EditColor.fdb 9KB FDB File 9/17/01 11:35 AM
ElementDistortion2D.fdb 3KB FDB File 9/17/01 11:35 AM
ElementDistortion3D.fdb 5KB FDB File 9/17/01 11:35 AM
Example4.fdb 47KB FDB File 9/17/01 11:35 AM
187 object(s) 49.2MB (Disk free space: 0 bytes) My Computer

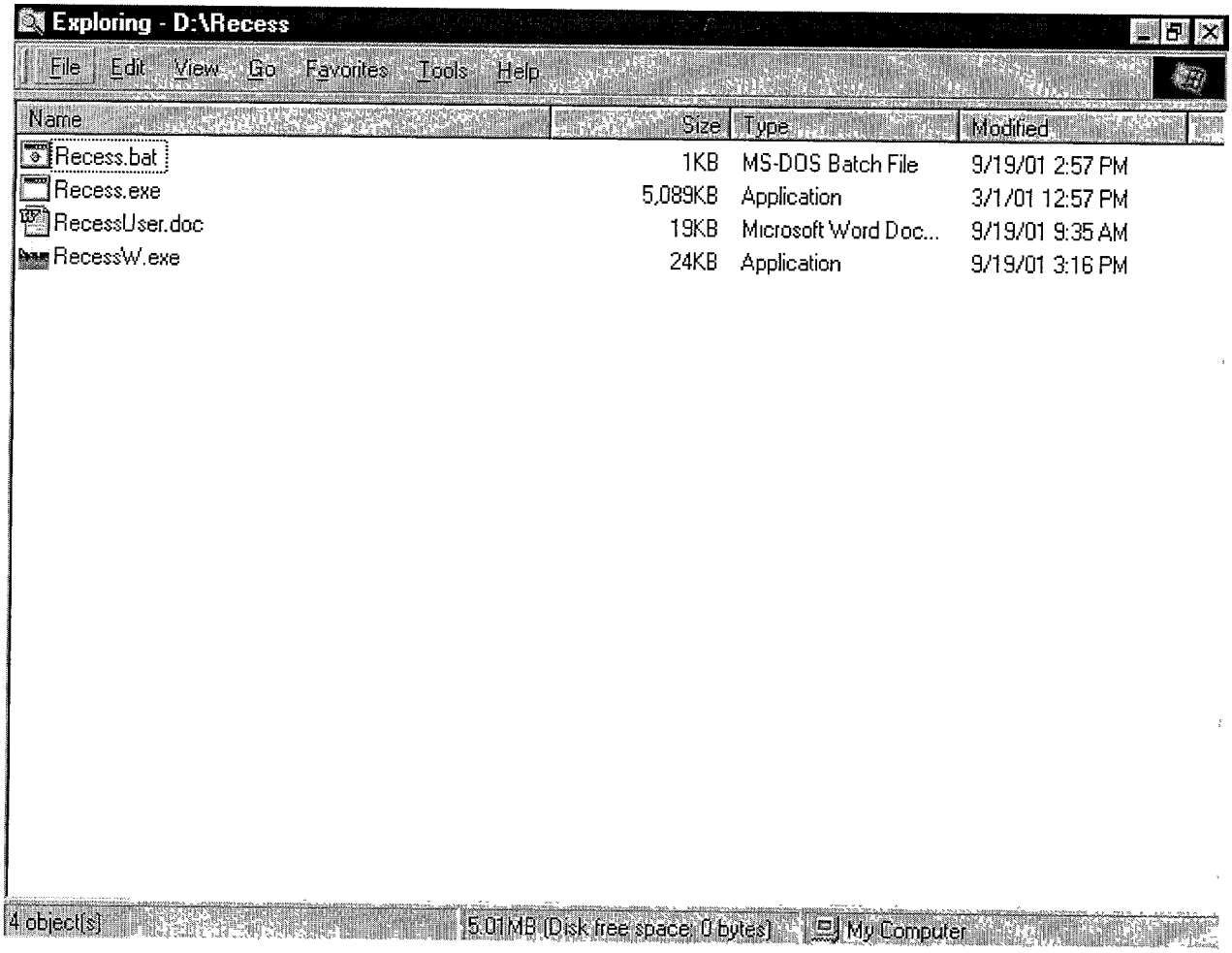

Exploring - D:\FEM Builder 1.2\Test
File Edit View Go Favorites Tools Help
Name Size Type Modified
Example4.fil 246KB FIL File 9/17/01 11:35 AM
Example4.inp 16KB INP File 9/17/01 11:35 AM
Example4F.fdb 33KB FDB File 9/17/01 11:35 AM
ExpCurve.dat 2KB Movie Clip 9/17/01 11:35 AM
fdv.nam 1KB NAM File 9/17/01 11:35 AM
fdv.xyz 5KB XYZ File 9/17/01 11:35 AM
fdv0001.dat 14KB Movie Clip 9/17/01 11:35 AM
fdv3d.nam 1KB NAM File 9/17/01 11:35 AM
fdv3d.xyz 28KB XYZ File 9/17/01 11:35 AM
fdv3d0001.dat 91KB Movie Clip 9/17/01 11:35 AM
FluidCFD1 fdb 8KB FDB File 9/17/01 11:35 AM
FluidCFD1.inp 3KB INP File 9/17/01 11:35 AM
Geometry.fdb 22KB FDB File 9/17/01 11:35 AM
GeometryEdit_1.fdb 3KB FDB File 9/17/01 11:35 AM
GridGen.grd 6KB GRD File 9/17/01 11:35 AM
GridGen.p3d 23KB P3D File 9/17/01 11:35 AM
Heat0.fdb 8KB FDB File 9/17/01 11:35 AM
Heat0.sot 11KB SOT File 9/17/01 11:35 AM
Heat1.fdb 9KB FDB File 9/17/01 11:35 AM
Heat1.sot 4KB SOT File 9/17/01 11:35 AM
HTAxisym1.fdb 8KB FDB File 9/17/01 11:35 AM
InsertFlawLinear.fdb 35KB FDB File 9/17/01 11:35 AM
InsertFlawQuadratic.fdb 69KB FDB File 9/17/01 11:35 AM
187 object(s) 49.2MB (Disk free space: 0 bytes) My Computer


Exploring - D:\FEM Builder 1.2\Test
File Edit View Go Favorites Tools Help
Name Size Type Modified
InterfaceFemDB_1.fdb 13KB FDB File 9/17/01 11:35 AM
InterfaceFemDB_2.FDB 15KB FDB File 9/17/01 11:35 AM
InterfaceIdeas_1.unv 31KB UNV File 9/17/01 11:35 AM
LessFluid.fdb 1,507KB FDB File 9/17/01 11:35 AM
LessFluid-CustomSolve.jou 1KB JOU File 9/17/01 11:35 AM
LessSolid.fdb 620KB FDB File 9/17/01 11:35 AM
LineElement.fdb 2KB FDB File 9/17/01 11:35 AM
MatProp1.pdb 2KB PDB File 9/17/01 11:35 AM
MatProp2.pdb 1KB PDB File 9/17/01 11:35 AM
MeshSurface.fdb 20KB FDB File 9/17/01 11:35 AM
MeshVolume.fdb 24KB FDB File 9/17/01 11:35 AM
MixingElbow.cas 129KB CAS File 9/17/01 11:35 AM
MixingElbow.dat 212KB Movie Clip 9/17/01 11:35 AM
MixingElbow.fdb 657KB FDB File 9/17/01 11:35 AM
Nozzle1.jou 1KB JOU File 9/17/01 11:35 AM
Nozzle1_ascii.cas 263KB CAS File 9/17/01 11:35 AM
Nozzle1_ascii.dat 980KB Movie Clip 9/17/01 11:35 AM
NozzleModel.cas 172KB CAS File 9/17/01 11:35 AM
NozzleModel.dat 365KB Movie Clip 9/17/01 11:35 AM
P&NRegions.fdb 4KB FDB File 9/17/01 11:35 AM
PipeFlow.cas 28KB CAS File 9/17/01 11:35 AM
PipeFlow.dat 11KB Movie Clip 9/17/01 11:35 AM
PipeFlow.fdb 7KB FDB File 9/17/01 11:35 AM
187 object(s) 49.2MB (Disk free space: 0 bytes) My Computer


Exploring - D:\FEM Builder 1.2\Test
File Edit View Go Favorites Tools Help
Name Size Type Modified
PlaneE1.fdb 8KB FDB File 9/17/01 11:35 AM
PlaneE1.fil 30KB FIL File 9/17/01 11:35 AM
PlaneE1.inp 3KB INP File 9/17/01 11:35 AM
PlaneE1.unv 12KB UNV File 9/17/01 11:35 AM
PlaneE1G.fdb 9KB FDB File 9/17/01 11:35 AM
PlaneE2.fdb 43KB FDB File 9/17/01 11:35 AM
PlaneE2.fil 223KB FIL File 9/17/01 11:35 AM
PlaneE2.resinp 16KB RESINP File 9/17/01 11:35 AM
PlaneE2F.fdb 152KB FDB File 9/17/01 11:35 AM
PlaneE2F.unv 159KB UNV File 9/17/01 11:35 AM
PlaneS1.fdb 10KB FDB File 9/17/01 11:35 AM
PlaneS2.fdb 65KB FDB File 9/17/01 11:35 AM
PlaneS2.fil 386KB FIL File 9/17/01 11:35 AM
RdAschar.plt 79KB PLT File 9/17/01 11:35 AM
RdCCM_1.ccmout 2KB CCMOUT File 9/17/01 11:35 AM
RdCDCA_1.cdcout 67KB CDCOUT File 9/17/01 11:35 AM
RWAnsys1.cdb 20KB CDB File 9/17/01 11:35 AM
RWAnsys2.cdb 7KB CDB File 9/17/01 11:35 AM
RWIdeas1.unv 185KB UNV File 9/17/01 11:35 AM
SBRAMSubScale.fdb 242KB FDB File 9/17/01 11:35 AM
SelectByRegion.fdb 4KB FDB File 9/17/01 11:35 AM
SelectCurve.fdb 14KB FDB File 9/17/01 11:35 AM
SelectGroup.fdb 15KB FDB File 9/17/01 11:35 AM
187 object(s)
49.2MB (Disk free space: 0 bytes)
My Computer


Exploring - D:\FEM Builder 1.2\Test
File Edit View Go Favorites Tools Help
Name Size Type Modified
SelectN,E,F.fdb 8KB FDB File 9/17/01 11:35 AM
SelectSurface.fdb 14KB FDB File 9/17/01 11:35 AM
SelectSurface1.fdb 14KB FDB File 9/17/01 11:35 AM
SelectVolume.fdb 14KB FDB File 9/17/01 11:35 AM
sharp.plt 6KB PLT File 9/17/01 11:35 AM
SinGen.gin 27KB GIN File 9/17/01 11:35 AM
SinGen.out 1,009KB OUT File 9/17/01 11:35 AM
Smooth2d_1.fdb 17KB FDB File 9/17/01 11:35 AM
Smooth2d_2.fdb 16KB FDB File 9/17/01 11:35 AM
Smooth2d_3.fdb 54KB FDB File 9/17/01 11:35 AM
Solid1L.fdb 9KB FDB File 9/17/01 11:35 AM
Solid1L.fil 53KB FIL File 9/17/01 11:35 AM
Solid1L.resinp 4KB RESINP File 9/17/01 11:35 AM
Solid1L.unv 13KB UNV File 9/17/01 11:35 AM
Solid1Q.fdb 20KB FDB File 9/17/01 11:35 AM
Solid1Q.fil 133KB FIL File 9/17/01 11:35 AM
Solid1Q.resinp 10KB RESINP File 9/17/01 11:35 AM
Solid1Q.unv 1KB UNV File 9/17/01 11:35 AM
SolidCFD1.fdb 9KB FDB File 9/17/01 11:35 AM
SolidCFD1.inp 3KB INP File 9/17/01 11:35 AM
StructuralGrid.fdb 35KB FDB File 9/17/01 11:35 AM
TexChem_1.fdb 9KB FDB File 9/17/01 11:35 AM
TexChem_1.spatial 13KB SPATIAL File 9/17/01 11:35 AM
187 object(s) 49.2MB (Disk free space: 0 bytes) My Computer


Exploring - D:\FEM Builder 1.2\Test
File Edit View Go Favorites Tools Help
Name Size Type Modified
sharp.plt 6KB PLT File 9/17/01 11:35 AM
SinGen.gin 27KB GIN File 9/17/01 11:35 AM
SinGen.out 1,009KB OUT File 9/17/01 11:35 AM
Smooth2d_1.fdb 17KB FDB File 9/17/01 11:35 AM
Smooth2d_2.fdb 16KB FDB File 9/17/01 11:35 AM
Smooth2d_3.fdb 54KB FDB File 9/17/01 11:35 AM
Solid1L.fdb 9KB FDB File 9/17/01 11:35 AM
Solid1L.fil 53KB FIL File 9/17/01 11:35 AM
Solid1L.resinp 4KB RESINP File 9/17/01 11:35 AM
Solid1L.unv 13KB UNV File 9/17/01 11:35 AM
Solid1Q.fdb 20KB FDB File 9/17/01 11:35 AM
Solid1Q.fil 133KB FIL File 9/17/01 11:35 AM
Solid1Q.resinp 10KB RESINP File 9/17/01 11:35 AM
Solid1Q.unv 1KB UNV File 9/17/01 11:35 AM
SolidCFD1.fdb 9KB FDB File 9/17/01 11:35 AM
SolidCFD1.inp 3KB INP File 9/17/01 11:35 AM
StructuralGrid.fdb 35KB FDB File 9/17/01 11:35 AM
TexChem_1.fdb 9KB FDB File 9/17/01 11:35 AM
TexChem_1.spatial 13KB SPATIAL File 9/17/01 11:35 AM
ThermalGrid.unv 8KB UNV File 9/17/01 11:35 AM
UnitCube.fdb 7KB FDB File 9/17/01 11:35 AM
UnitSquare.fdb 3KB FDB File 9/17/01 11:35 AM
187 object(s)
49.2MB (Disk free space: 0 bytes)
My Computer



Case 3:01-cv-00988-SI Document 203-3 Filed 04/05/05 Page 1 of 1